What's new in NX 10.0.1

Contents

Chapter 1: Teamcenter Integration for NX

Table editor layout

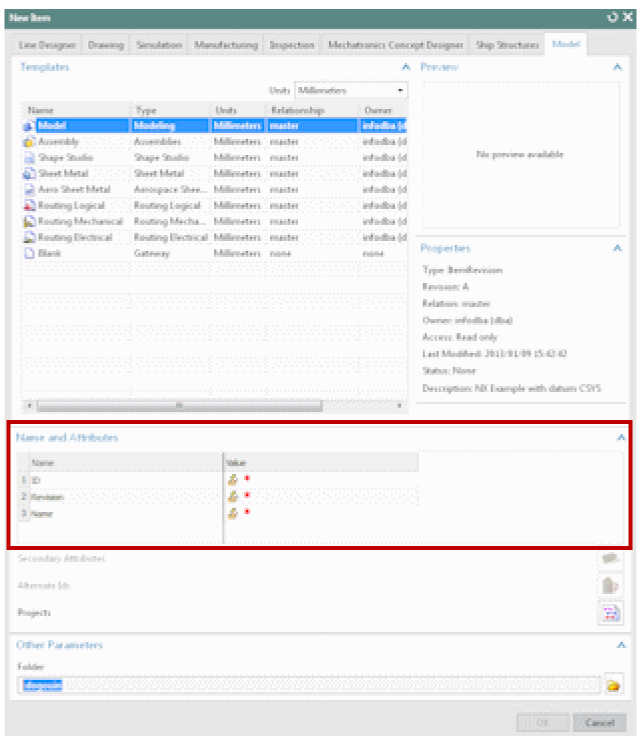
What is it?

A table editor layout is now incorporated into commands and dialog boxes in the user interface. This provides a quicker use of commands with fewer mouse clicks and easier use of dialog boxes when you have multiple objects involved in the operation.

The table editor layout provides a spreadsheet style of user input so you can input values for multiple items without *leaving* an item in the dialog box to proceed to the next one. You can also input values for multiple attributes in a cohesive table of attributes and values.

The following are some examples of the table editor layout:

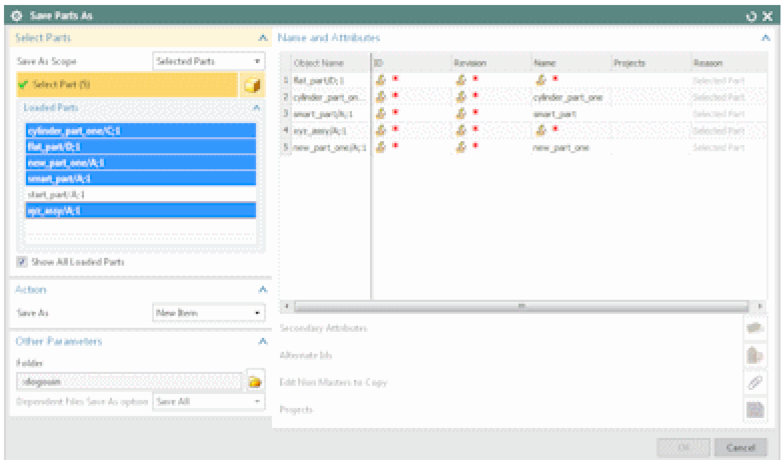
- **File** tab→**New**→**Item**



  - o The red asterisk indicates the value is required.
  - o You can highlight the cell and enter a value, or double-click to get the insert mode and type in the cell.
  - o If the cell has auto-assign , you can double-click to automatically assign a value.

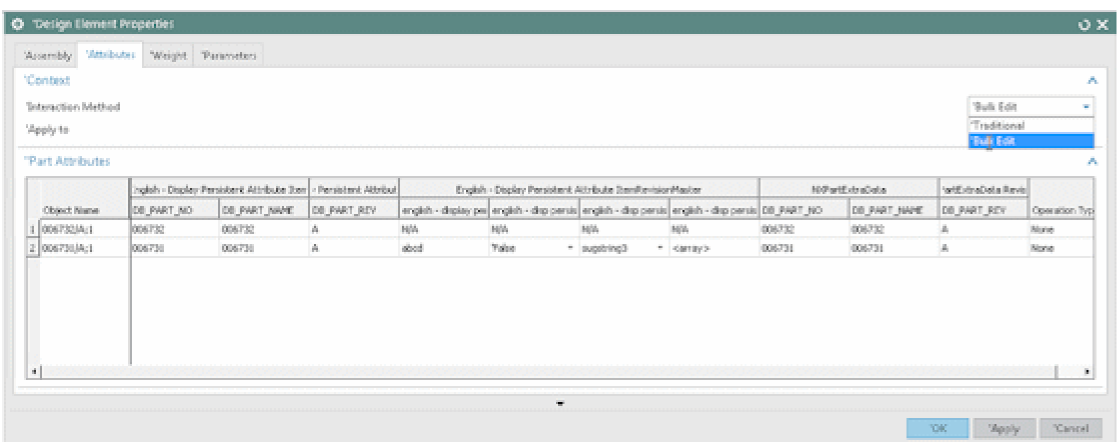
- You can right-click and use the **Edit** dialog box to add or change an attribute.

- **File** tab→**Save**→**Save As**



  - You can filter a column listing to display parts based on your selection criteria.

- Bulk edit of attributes (example from **Design Element Properties** dialog box)



Note

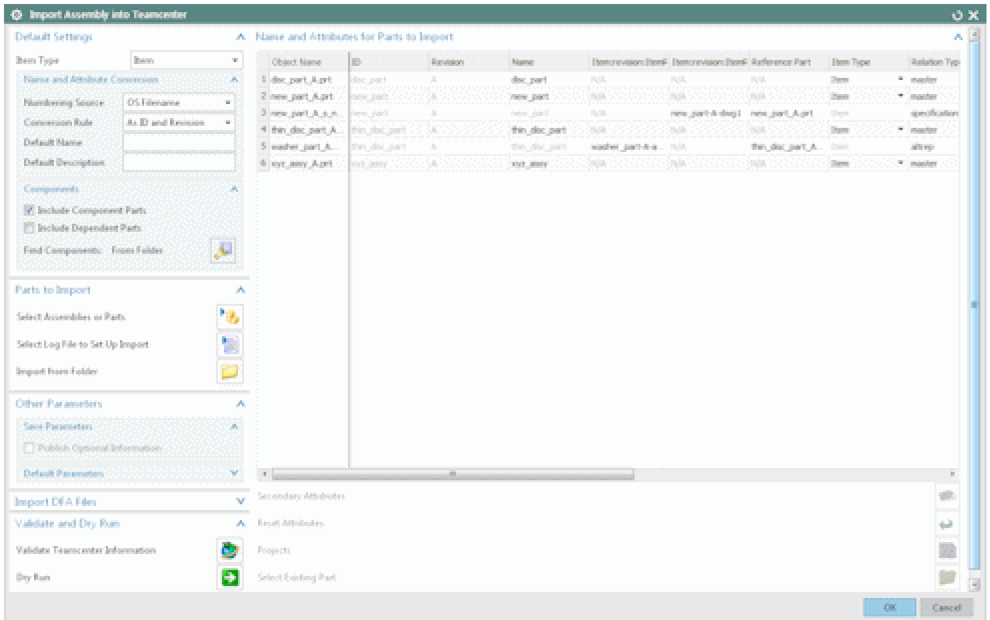
The table editor layout for attributes is applicable when you are running in Teamcenter Integration and native NX.

  - Use the **Bulk Edit** view to get the table editor layout.

  - For items that have the same attribute, you can select the cells in the attribute column for the items and make the same edit for all of the items.

  - You can hide a column and decrease the size of the table.

- **File** tab→**Import Assembly into Teamcenter**



  - o The table editor layout is applicable for single or multiple items.

Why should I use it?

You can more easily enter values for multiple items and multiple attributes.

Where do I find it?

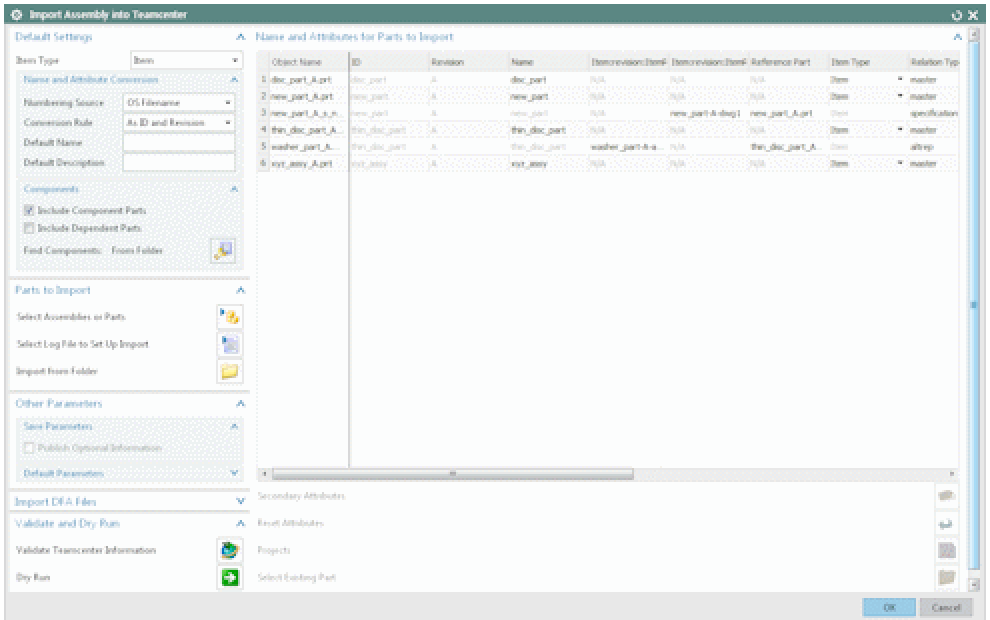
Application	Applies to all applications when using **Teamcenter Integration**. Table editor layout for bulk edit of attributes also applies to native NX.

Import assembly into Teamcenter

What is it?

The **Import Assembly into Teamcenter** command is redesigned to provide improved performance, an easier-to-use interface, and added functionality.

The new interface eliminates multiple tabs and includes all of the options in the dialog box on one page. The existing functionality for the command is preserved, and in addition you can now list multiple objects at once with their associated attributes in a spreadsheet style format.



Additional options are added to support the new import functionality in the **Import** tab in the Teamcenter Integration for NX customer defaults.

Also, a new command-line utility, **tcin_import**, is added that incorporates the options and features of the redesign.

Note

The new **Import Assembly into Teamcenter** command can be more memory intensive when importing parts with the **Select Log File to Set Up Import** and **Import from Folder** options. More memory is also used when the **Publish Optional Information** option is selected. If memory is a concern, the old **Import Assembly into Teamcenter (legacy)** command is hidden but can be found by using **Command Finder**.

Why should I use it?

You can more easily import a part or assembly into Teamcenter from the operating system.

Where do I find it?

Application	Teamcenter Integration
Ribbon bar	**File** tab→**Import Assembly into Teamcenter**

UTF-8 encoding supported

What is it?

UTF-8 character encoding is now always supported in NX by default. NX internally stores all data in the UTF-8 character encoding and the NX user interface supports UTF-8. This lets you author NX data in any supported language and view it correctly in any another supported language. Previously, NX was running character encoding as determined by the operating system.

You can enter data in NX and have it stored in Teamcenter regardless of the language of the character set that is used, as long as character encoding for the character set is supported by Teamcenter.

The Teamcenter setup may or may not support UFT-8 character encoding, which can cause issues if NX supports encoding of a character set and Teamcenter does not. In these cases, an error message notifies you that the character set is not supported by Teamcenter.

If you are running NX against a Teamcenter database that supports UTF-8 character encoding, all characters that are supported by UTF-8 are valid. If you are running NX against a non-UTF-8 Teamcenter database, then validation occurs before NX accepts the characters. At key locations in NX, user input is immediately evaluated as being valid input for the character encoding that is supported by your Teamcenter installation. At other locations, user input is evaluated when you perform a **Save** or **Save As** on the item.

There are many places within NX where data is input, so only the most common or key data input locations are immediately evaluated. For example, in the **New Item** dialog box, the **ID**, **Name**, and **Revision**, boxes are immediately evaluated for language compliance.

Why should I use it?

You can enter data in NX regardless of the language and have it stored correctly in Teamcenter, especially multi-byte characters, provided that Teamcenter supports the character encoding.

Where do I find it?

Application	Applies to all applications when using **Teamcenter Integration**.

Using Teamcenter multifield keys

What is it?

Teamcenter multifield keys are typically used for non-design data such as documents and are rarely used for the design data itself. When there are many types of documents associated with a design, multifield keys provides a convenient way in Teamcenter to assign the same item ID to all of the various documents while using a second field to differentiate between the types of documents. It is the combination of a Teamcenter's object item ID and the item type that provides the unique key in Teamcenter for identifying a particular document associated with a design's item ID. For example, the multifield key can be defined as **Item ID, Item Type** where the item type could be requirements, certification, data sheet, quality tests, analysis results, and so on.

For design data, each design item typically represents a unique component in the product, which in turn must be uniquely identified in downstream configurations by a single key, such as manufacturing systems, procurement systems, ERP, and so on. Therefore, using multifield keys for design items poses definite risks to end-to-end business processes.

Using the Teamcenter concept of domains, you can have design data exist in a domain where only a single key (item ID) is used to identify uniqueness. And you can have a second domain for document data where a multifield key (item ID, item type) is used to identify uniqueness. By using this method, design data cannot become ambiguous or pose risks to downstream systems, while all document data can be managed using the same item ID as the design data to which it is associated.

Caution

Using multifield keys for design data should be carefully considered prior to actual use within a production environment. Prior to implementing multifield keys you should assess your business processes and take every step to remove the risk of introducing ambiguity into the way design data is identified and managed.

Work with objects in all Teamcenter MFK domains

What is it?

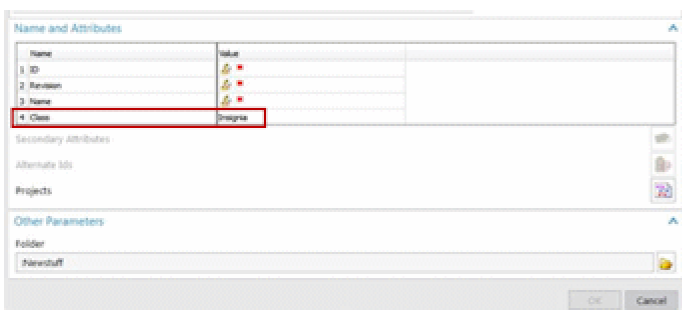
NX utilizes Teamcenter multifield key (MFK) functionality so that you can now view and work with objects that are in all Teamcenter domains, not just the default domain.

The Teamcenter MFK is a set of key attributes that serve as the unique identifier for an item object. These can include item_id, object_name, supplier_code, and so on. The Teamcenter **Part Identifier** property on an object contains the names and values of all key properties for the MFK. The NX part attribute **DB_PART_MFKID** corresponds to the Teamcenter **Part Identifier** property and is automatically added to all parts as a hard-coded attribute.

In Teamcenter, you can add customizations which include properties and creation descriptors to the item type in the BMIDE so that you can further identify and differentiate parts to utilize the MFK functionality. These customizations, or key properties, comprise the MFK.

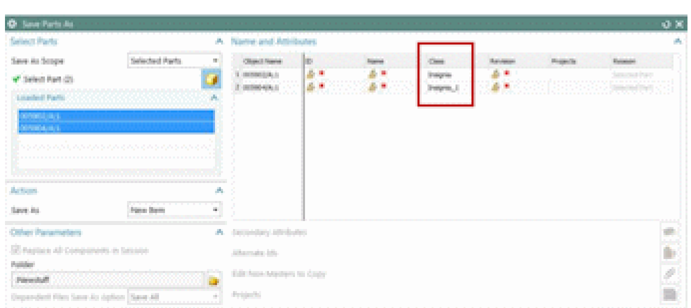
The required key properties are shown in NX dialog boxes as attributes when you create a new item. The following are examples where you would have to add values for additional attributes for MFK:

- **File** tab→**New**→**Item**



  - o In the **Name and Attributes** group, another key attribute (with a default value) is added in addition to **ID**, **Revision**, and **Name**.

- **File** tab→**Save**→**Save As**



- o In the **Name and Attributes** group, another key attribute column is added in addition to **ID**, **Name**, **Revision**, **Projects**, and **Reason**.

Command line programs, such as utilities, are also updated to accept the Teamcenter **Part Identifier** (contains the MFK) as input.

You can configure NX so that NX data is only in the Teamcenter default domain where all NX items have only item_id as the key attribute. In this configuration, NX accepts the item ID as the unique identifier and NX runs as it did prior to NX 10.

Why should I use it?

You can use the Teamcenter MFK functionality to more specifically identify an item with a domain in Teamcenter.

Where do I find it?

Prerequisite	Your site is setup to utilize the MFK functionality.

MFK for part families

What is it?

Teamcenter multifield key functionality (MFK) is now supported by NX and the part family functionality also supports MFK. Any Teamcenter customization attributes or creation descriptors created for MFK are added in the **Part Families** dialog box and spreadsheet. This lets you easily find which attributes are required for part creation.

In the **Part Families** dialog box, the MFK attributes are added and identified as required for part creation in the new **Category** column. As with the existing required attributes, the attributes specific to MFK cannot be deleted, reordered, or copy/pasted.

In the spreadsheet, when you use item names for components and materials, you specify the attributes of the MFK with a user-friendly format. For example, for an MFK specified as: ”%#MFK#%,=mfk9Const=C,item_id=MFKC2–AA00054,object_type=MFK9C2T1”, you enter it as (where *Alt+Enter* is the keyboard entry):

- mkf9Const=C *Alt+Enter*
- item_id=MFKC2–AA00054 *Alt+Enter*
- object_type=MFK9C2T1

In addition, part attributes mapped from Teamcenter can now have both internal names and display names. If an attribute has different display and internal names, in the spreadsheet the part family column is changed to show **display_name(internal_name)**. The same combination is also shown for the attribute in the **Available Columns** and **Chosen Columns** list.

Why should I use it?

You can ensure that all part family members have the required MFK attributes.

Where do I find it?

Application	Teamcenter Integration
Dialog box	**Part Families**

Operation descriptors

What is it?

In Teamcenter, you can add operation descriptors to a business object in the BMIDE, such as Item, Item Revision, Item Master Form, and so on, to let you customize the inputs that are collected based on the operation performed, such as creating a new part or performing a Save As on an existing part.

When you add Teamcenter operation descriptors, these are now reflected in NX where you need to input values for these properties when you are creating an object. Teamcenter can designate operation descriptors as required or not required, but NX displays only the descriptors that are required.

The operation descriptors are shown as additional attributes and are in addition to the typical required attributes. For example, when you create a new item, you could have the following required attributes:

ID (typical required attribute)

Revision (typical required attribute)

Name (typical required attribute)

Make or Buy (operation descriptor)

Why should I use it?

You can use operation descriptors to require values for additional attributes for specific NX operations.

Where do I find it?

Prerequisite	You must be creating a new object or performing a Save As.

Active Workspace data synchronization

What is it?

Active Workspace is enhanced to support data synchronization between NX and **Active Workspace** when you are using **Active Workspace** in NX.

Data synchronization ensures that when object level information for a part, assembly, or other item is modified in NX, **Active Workspace** is automatically refreshed to show the change and vice versa. For example, if you check out a part in **Active Workspace**, it is shown as checked out in NX, or if you add a component to an assembly in NX, **Active Workspace** refreshes and the BOM for the assembly is automatically updated.

Why should I use it?

When you use **Active Workspace** in NX, you are provided with more up-to-date and accurate information.

Where do I find it?

Application	**Active Workspace** in NX

Reconnect to Teamcenter

What is it?

If you lose your connection to Teamcenter, NX automatically reconnects to enable you to continue working. A disconnection can occur due to an idle session, a timeout policy at your site, a server that goes down due to network problems, or other issues.

A message is displayed when you are reconnected. You have the option of continuing the current session or exiting the session. If you continue, depending on the last NX command you initiated, the command may have not completed before you were disconnected and you may have to run that command again.

This functionality is available only when you are running Teamcenter 4–tier deployment architecture.

In addition, depending on the setup at your site, NX may attempt to connect to a new Teamcenter server, especially if you lost connection due to a network problem.

Why should I use it?

You can automatically reconnect to a Teamcenter server if connection is lost.

Display names

What is it?

The display names (not the internal names) that are defined for properties in Teamcenter are used in NX. The properties are defined in the Teamcenter BMIDE and shown in NX as attributes on dialog boxes. Previously, the attribute names used in NX were determined by the Teamcenter mapping file definition.

This provides a consistent naming between properties in Teamcenter and the corresponding attributes in NX. The most noticable attributes affected are **ID** and **Revision** for item ID and item revision as these are used throughout NX. All dialog boxes that contain attributes are affected, such as **New Item**, **Save Parts As**, and **Import Assembly into Teamcenter**.

In addition, the attribute names in NX are localized if the localization language is supported.

Why should I use it?

You can more easily locate similar attributes in Teamcenter and NX due to the consistent display of attribute names.

Localizable attributes

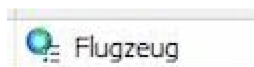
What is it?

When you are running NX in a language other than the master locale (the main language used for data input in the Teamcenter database), the translation available  and no translation available  icons are displayed to identify whether your input can be translated into another language (localized). These translation icons are displayed only when you are using the bulk edit layout of attributes.

Attributes are defined in Teamcenter and used in NX. In Teamcenter, you can define an attribute as localizable, which lets your Teamcenter administrator enter localized text for the attribute values (string values only). You could have several localized values for an attribute depending on the number of languages supported. The attributes that are localizable and the languages supported are determined by the requirements at your site.

If you are running NX in a language that is the same as the master locale, or is different than the master locale but the attribute is not localizable, then a translation icon is not displayed.

If you are running NX in a language that is different than the master locale and localized text is available for the input value, then the translation available icon is displayed next to the value.



If localized text is not available for the attribute, then the no translation available icon is displayed next to the value and the value is shown in the master locale language.

Why should I use it?

You can see whether an attribute value is localizable.

Where do I find it?

Application	Applies to all applications when using Teamcenter Integration.

Drawing booklets in Teamcenter

Configuring Item Types for a Booklet Rule part

What is it?

You can now specify the Teamcenter item type that is used to create a drawing booklet.

Why should I use it?

Previously, a Teamcenter drawing booklet always created an item in the Teamcenter database using the **Item** type for the **Booklet Rule**. This could cause problems if you tried to create a drawing booklet while using a customized Teamcenter database where the **Item** type was discouraged or unavailable. Now that you can specify which item type is used, you can create drawing booklets that comply with your database rules.

Where do I find it?

Application	Drafting
Command Finder	**Customer Defaults**
Location in dialog box	**Drafting→Drawing Automation→General** tab→**Item Types** group→**Booklet Rule**

Configuring revisions for drawing parts in drawing booklets

What is it?

When you edit a drawing booklet, you can now designate the Teamcenter revision for the drawing parts.

If the part used as the basis for the booklet has the part attribute DRAWING_BOOKLET_ITEM_REVISION, its setting is assigned to the following:

- A new booklet and its drawings.
- New drawings that are created when an existing booklet is edited.

If the attribute DRAWING_BOOKLET_ITEM_REVISION is not used:

- Teamcenter revision rules determine the default revision.
- The assigned revision matches the revision number of the drawing booklet.

Where do I find it?

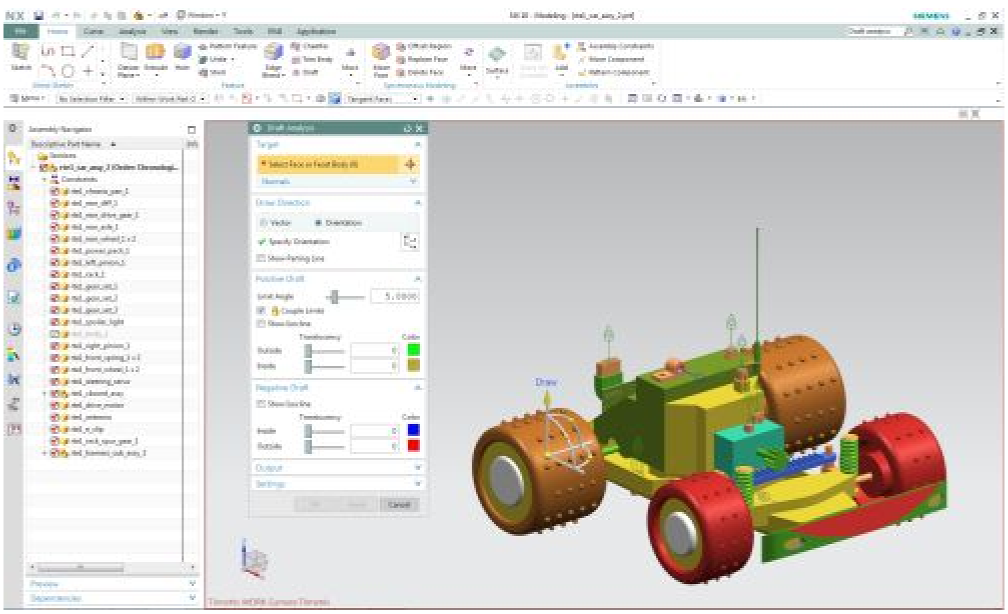
Application	Drafting
Menu	**Menu→Tools→Drawing Automation→Edit Drawing Booklet** **Menu→Tools→Drawing Automation→Create Drawing Booklet**

Chapter 2: Fundamentals

New NX look

What is it?

Siemens PLM Software products are adopting a consistent look. This look is now the default for the NX user interface.



Theme = Light

In NX, the **Theme** preference for the user interface layout controls the look.

You can use the following new preferences to set the theme:

- **Light**

  This is the default preference.

- **Light Gray**

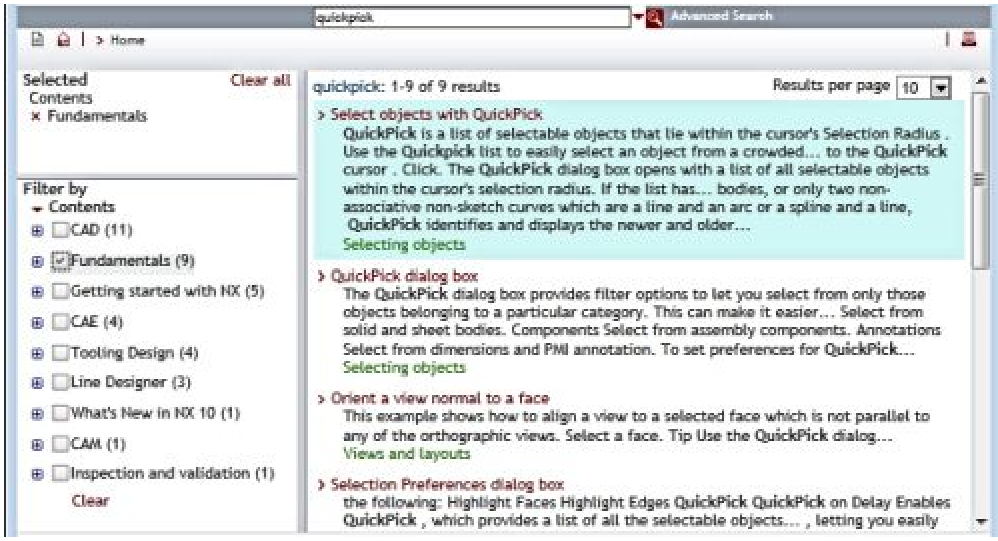
Where do I find it?

Command Finder	**User Interface Preferences**
Location in dialog box	**Theme** node→**NX Theme** group→**Type** list→**Light** or **Light Gray**

Help and search enhancements

What is it?

The NX Help is provided as a web application which allows you to more easily serve a single copy of the help to multiple users and also includes an improved web-based Search.



Benefits include:

- NX Help Search no longer requires Java applets and no longer breaks due to Java security updates.
- The Search is faster.
- Filtering is more intuitive.
  - o The Search sorts the results by content area.
  - o The **Filter by** pane displays the number of results found in each content area. Select one or more content areas to filter the results.
  - o The **Selected** pane lists the filters you select. In this pane, you may deselect one or more filters or clear all your filters.
- One copy of the help can be served to multiple users.

Note

We recommend you install and serve the NX Help to multiple users via an intranet. You can install the NX Help locally on individual workstations but this requires the additional installation of a web service to run the search on each local system and also incurs additional maintenance to each workstation if a newer version of the help is made available.

Where do I find it?

Command Finder	**Help** 

Resource bar enhancements

What is it?

You can now use the Resource Bar Options menu to control the location, display behavior, and content of the Resource bar. NX displays the Resource Bar Options menu on the top left corner of the Resource bar with the following options.

Pin Open	Turns on or off the auto-hide behavior of the navigators or palettes. This option replaces the and buttons in pre-NX 10 versions.
Maximize tab	Maximizes or restores the selected navigator or palette. The maximized navigator or palette covers the graphics window.
Left	Positions the Resource bar to the left of the graphics window.
Right	Positions the Resource bar to the right of the graphics window.
Overlap on Second Monitor	Positions the Resource bar on a second monitor at the edge nearest the NX window. If you maximize an application on the second monitor, the Resource bar is overlapped.
Dock on Second Monitor	Docks the Resource bar on a second monitor in a reserved space at the edge nearest the NX window. Applications that you maximize on the second monitor will not overlap the Resource bar.
Tabs	Lets you hide or display the selected tab.

Tip

To hide a tab or undock a navigator or palette, right-click the tab and choose the option you want. To show a hidden tab, right-click in an empty space on the Resource bar, and select the tab.

Customize line widths

You can customize the line widths in NX using customer defaults and user preferences.

You can:

- Specify labels for each line width in NX.
- Set the pixel width to display the corresponding new line width in NX. You can also set multiple lines with the same pixel width.
- Map the legacy line widths such as Thin, Normal, and Thick to the current line widths in NX.
- Set the scale factor for the line widths for printing.
- Define labels and pixel widths for the line for plotting and for PDF export.

Note

- You must map the legacy line widths in a logical way. For example, the line width to which you map a Thin width must be lower than the line width to which you map a Normal width. NX ignores invalid line width mapping and reports the details in the system log.

- To ensure that NX uses the new line widths in the newly created parts, you must update or recreate the templates with the new line widths.

Where do I find it?

Command Finder	**Customer Defaults**
Location in dialog box	• Defining line width labels **Gateway→Line Widths→Presentation** tab, **Width Names** group • Defining line width pixels **Gateway→Visualization→Line** tab, **Pixel Widths** group • Mapping legacy line widths **Gateway→Line Widths→Conversion** tab, **Legacy Width Conversion** group

Defining line width pixels using user preference

Command Finder	**Visualization Preference**
Location in dialog box	**Line** tab, **Pixel widths** group

Defining legacy line width mapping using user preference

Command Finder	**Object Preferences**
Location in dialog box	**Line Widths** tab, **Legacy Width Conversion** group

Creating output geometry for measurement features

What is it?

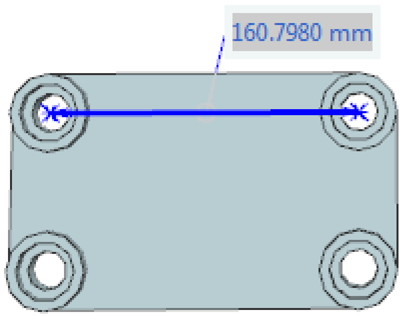
You can now create associative geometry for measurement features.

To do this, use the **Create Output Geometry** option for the following commands:

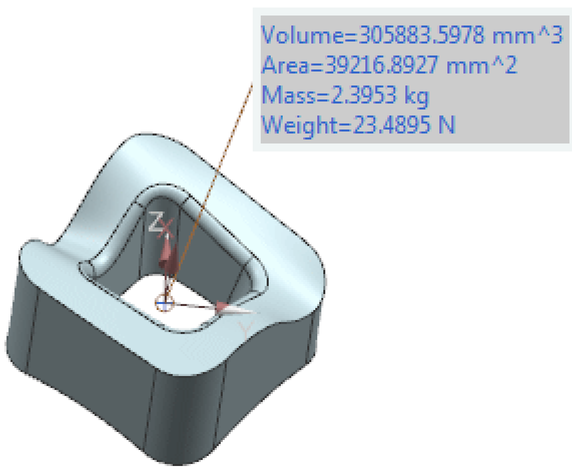
- **Measure Distance**
- **Measure Angle**
- **Measure Extremes**

For the **Measure Bodies** command, use the **Create Principal Axes** option.

Example



A line and two points created as an output of the distance measurement feature.



A point and a coordinate system created as an output of the body measurement feature.

Why should I use it?

Use the associative geometry to:

- Get visual feedback for the measurement features.
- Make associative references to the output geometry.

Where do I find it?

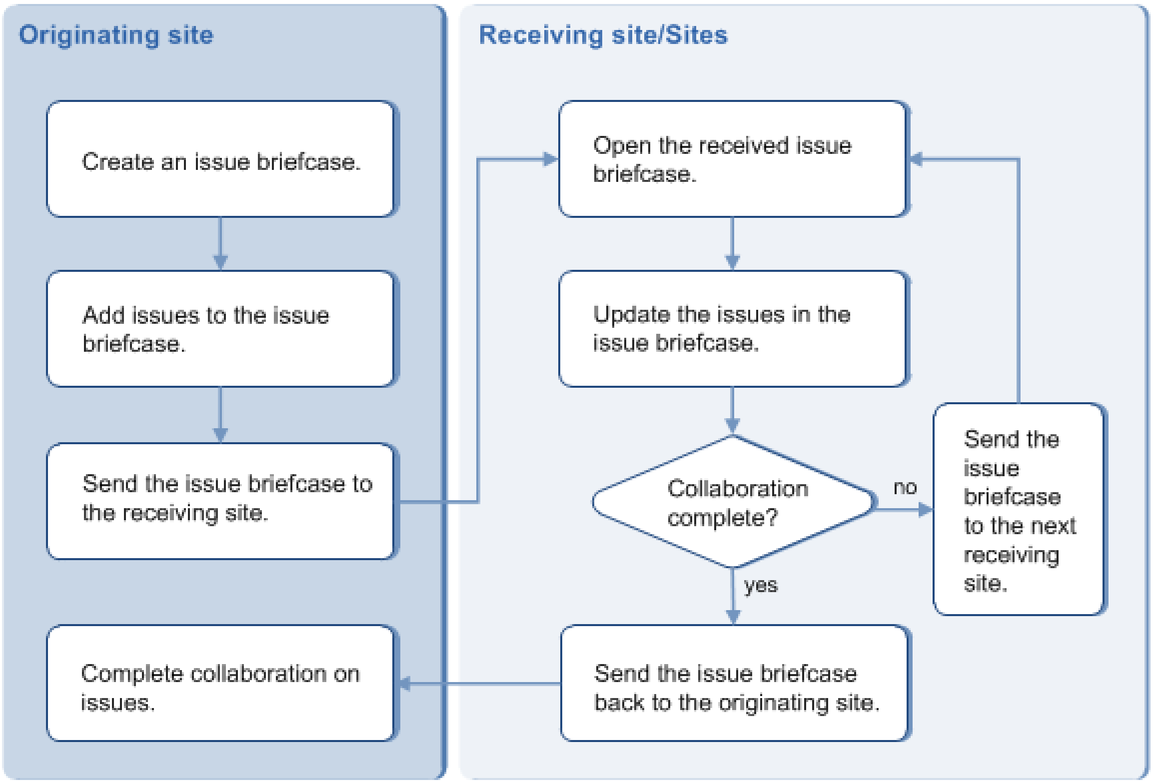
Prerequisite	All command dialog boxes: In the **Associative Measure and Checking** group, you must select the **Associative** check box. **Measure Distance** dialog box: Set **Type** to one of the following: • **Distance** • **Between Object Sets**

	• **Projected Distance** • **Projected Distance Between Object Sets** **Measure Angle** dialog box: Set **Type** to : • **By 3 Points**
Command Finder	**Measure Distance**, **Measure Angle**, **Measure Bodies**, **Measure Extremes**
Location in dialog box	All commands except **Measure Bodies**: **Results Display** group→**Create Output Geometry** **Measure Bodies**: **Results Display** group→**Create Principal Axes**

Collaborating on NX issues

NX teams at different sites can collaborate on issues using an issue briefcase, regardless of whether the team members work in native NX or Teamcenter Integration. An issue briefcase package file (.ibp) contains multiple issues and their attachments.

A typical workflow for collaboration on issues is as follows.

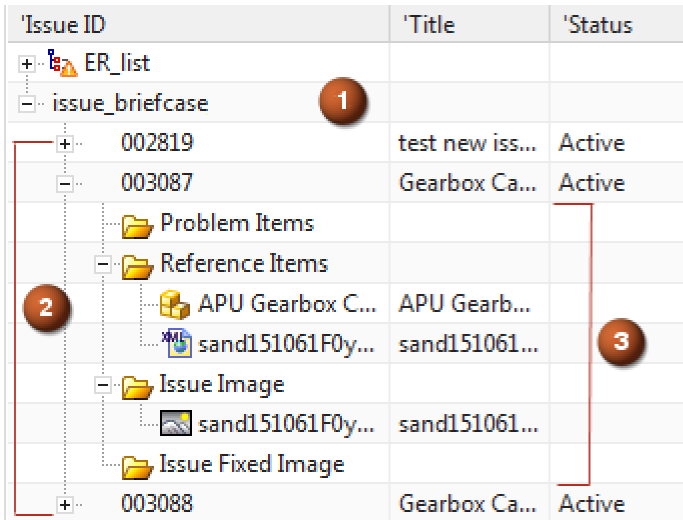


Example

An Original Equipment Manufacturer (OEM) site with Teamcenter or Teamcenter Community can use an issue briefcase to collaborate on issues with a supplier who does not have Teamcenter or Teamcenter Community issue management facilities. For multiple suppliers, the OEM site can create multiple issue briefcases to collaborate on issues.

You can create and open an issue briefcase, and also modify the issues in an issue briefcase without logging in to Teamcenter or Teamcenter Community. However, you must log in to Teamcenter or Teamcenter Community to add issues to the briefcase. If the originating site uses native NX, you can create the issues directly in the issue briefcase.

When you create an issue briefcase and add issues to it, or open a received briefcase, NX creates a node and sub-nodes in the Issue Management tool tree to list the issue briefcase and its content.



1	Issue briefcase
2	Issues added to the issue briefcase
3	Issue attachments

Note

- Teams at different sites cannot open an issue briefcase at the same time.
- Multiple users at a site cannot work on an issue briefcase at the same time.
- Team members at the receiving site cannot add or remove issues.
- You cannot collaborate on issues in 4GD.
- You cannot attach a bookmark or a snapshot of an issue in the issue briefcase.

Where do I find it?

Resource bar	**HD3D Tools** →**Issue Management**
Location in dialog box	**Create Issue Briefcase** **Open Issue Briefcase**

Hosting custom windows in NX

What is it?

You can now create custom Resource bar tabs to host your own windows. Use the following new NX Open APIs, which are in the **ResourceBarManager** class, to define and control the window.

Create
Creates a tab, which is added to the Resource bar when you set the visibility.

RegisterActivationCallback
Activates and displays the custom window when you click the tab.

ActivateTab
Activates and displays the custom window immediately; you do not need to click the tab.

GetWindowHandle
Creates an NX docking window, which is used as the parent to host your custom window.

SetTabAvailability
Controls the visibility of the tab in the Resource bar area.

Destroy
Removes your custom window and tab from NX.

You can specify a title and bitmap for your custom tab. Your custom window has standard Resource bar capabilities, such as the ability to dock and undock the window.

Note

The application that is being hosted controls the behavior inside the host window.

Why should I use it?

Use the APIs to seamlessly integrate third-party applications and dialog boxes into NX. For example, if you have your own add-in application or navigator that you use in conjunction with NX, you no longer need a separate window.

Where do I find it?

The new NX Open APIs are in the new **ResourceBarManager** class.

Zoom operation enhancements

What is it?

The default direction in which to scroll the mouse wheel to zoom in and zoom out, is changed.

- To zoom in, scroll forward.

  You can also hold the left and middle mouse buttons or hold Ctrl and the middle mouse button and move the mouse forward to zoom in.

- To zoom out, scroll backward.

  You can also hold the left and middle mouse buttons or hold Ctrl and the middle mouse button and move the mouse backward to zoom out.

To change the default scroll and drag direction, use the **Mouse Wheel Scrolling** customer default. This customer default is available only in NX running on Windows or the Linux operating system.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

Customize dialog box enhancements

What is it?

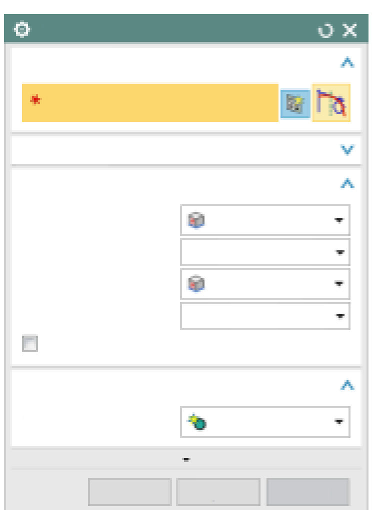
The following enhancements make it easier for you to locate commands that you want to add to the Ribbon bar:

- All NX commands are available under the new **All Commands** node in the **Categories** list on the **Commands** tab.

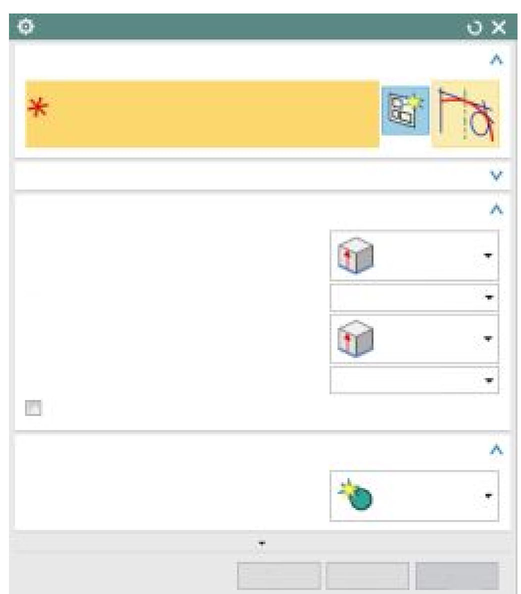
  To find an item under a category node, select the node, and enter your search term in the **Search** box.

- All the Ribbon bar drop-down lists are available under the **All Dropdowns** node.

You can change the size of icons in dialog boxes. To do this, on the **Icon/Tooltips** tab, under **Icon Sizes**, select an option from the **Dialog Boxes** list.



Dialog Boxes = Normal



Dialog Boxes = Large (200%)

Usability enhancements include the following:

- The **Layout** and **Roles** tabs are removed. These options are now available in the **User Interface Preferences** dialog box.
- The **Options** tab is now called **Icon/Tooltips**.
- The new **Tabs/Bars** tab lists all the tabs and bars in the NX interface. To show or hide a tab or a bar, select or clear the check box next to its name in the list.

Where do I find it?

Command Finder	**Customize**
Location in dialog box	**Customize** dialog box→**Commands** tab→**Search** **Customize** dialog box→**Icons/Tooltips** tab→**Dialog Boxes**

Ribbon bar enhancements

What is it?

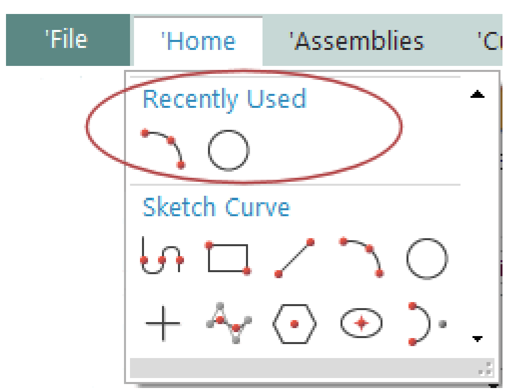
You can customize the Ribbon bar in the following ways when the **Customize** dialog box is open. To open this dialog box, right-click the Ribbon bar and choose **Customize**.

- You can control the display of command buttons using enhanced command button display styles.

  To do this, right-click a command and choose a style from a size style option.

  To return to the default display, use the **Reset** option on the same menu.

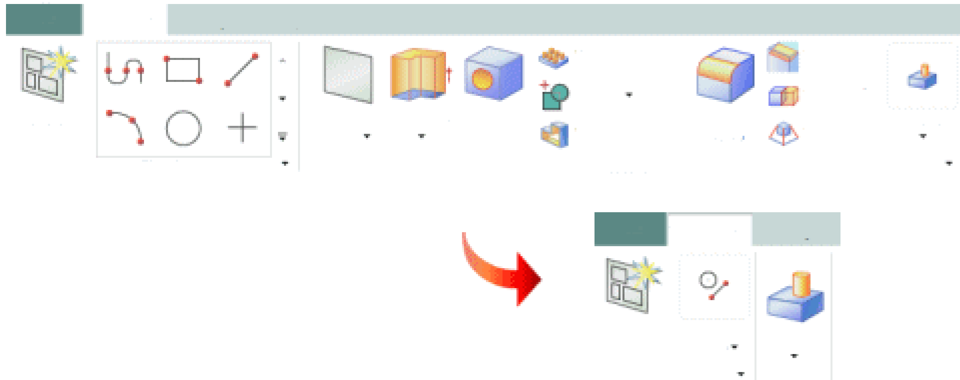
- You can track and reuse recently used commands in a gallery.

  To do this, right-click a gallery and choose **Recently Used**→**Enable Recently Used**.

- You can collapse a group or gallery.

  To do this, right-click the name of the group or gallery and choose **Collapsed**.



- You can add new tabs using the  button.

  To add a tab, click this button and add the name you want in the **Name** box in the **Tab Properties** dialog box.

You can also undock the Ribbon bar or hide tabs on the Ribbon bar. The **Customize** dialog box need not be open when you do this.

- To undock the Ribbon bar, right-click it and choose **Undock Ribbon**.

- To hide tabs, right-click the tab name and choose **Hide Tab**.

Cue and Status line enhancements

What is it?

You can now control the visibility of the Cue and Status lines. To do this, right-click the Ribbon bar and choose **Cue/Status Line**.

By default, the Cue and Status lines are not displayed in Full Screen mode. To turn on its display, right-click the Ribbon bar and choose **Cue/Status Line**.

User Interface Preferences enhancements

What is it?

You can now access all the interface preferences from the **User Interface Preferences** dialog box. These preferences are reorganized into logical groups.

To locate a specific user interface preference quickly, use the **Find**  option.

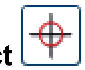
Where do I find it?

Application	Gateway
Command Finder	**User Interface Preferences**

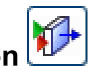
View Section enhancements

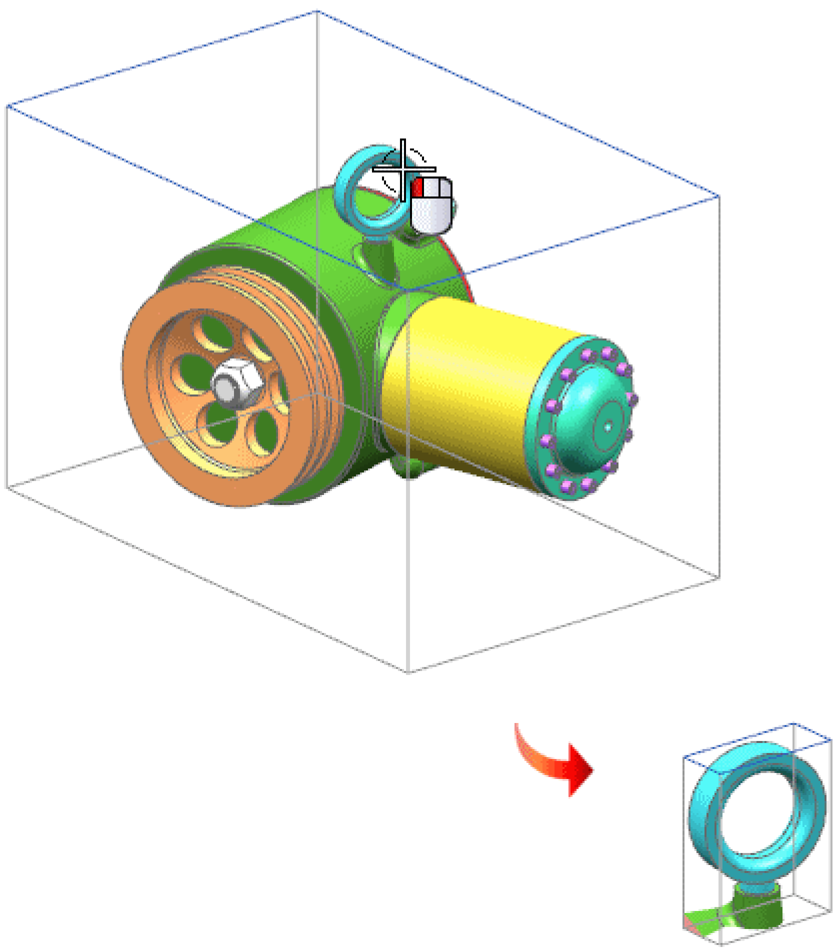
What is it?

When you create or edit a section view, you can specify the location and extent of the section box by selecting one or more objects and control the size of the section box by setting a margin. NX applies the margin to all sides of the section box. NX automatically updates the display of the section box unless you delay the update.

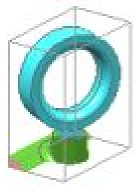
- To select one or more objects, use the **Select Object**  option in the **Extent** group or select the component in the **Assembly Navigator**.

  Tip

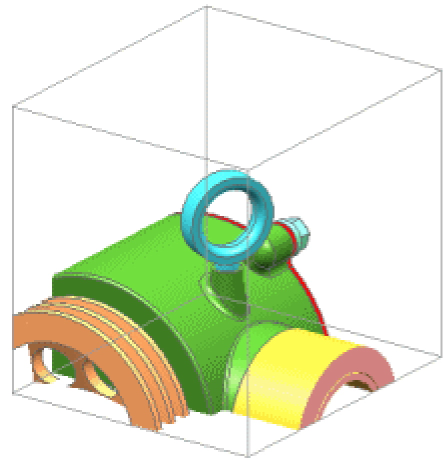
  To select an object outside the box, turn off **Clip Section**  on the Top Border bar.



- To specify a margin, enter a value in the **Margin** box.



Margin = 0



Margin = 3

The **Box Margin** customer default controls the default margin value.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

- To stop the dynamic update of the section box, select the **Delay Box Update** check box. If you want to update the section box when the **Delay Box Update** check box is selected, click **Update Box** .

Why should I use it?

In a large assembly, you can quickly define the section areas by creating section boxes around the selected components.

Where do I find it?

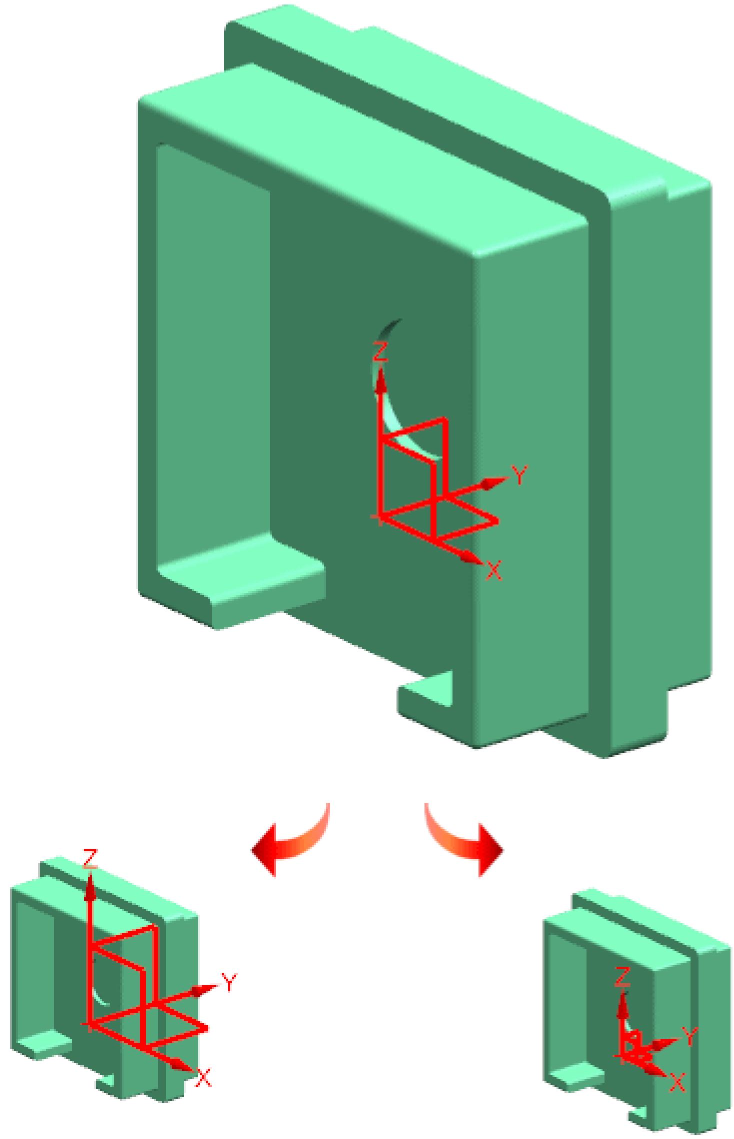
Command Finder	**New Section** **Edit Section**
Location in dialog box	**View Section** dialog box→**Extent** group

CSYS and View Triad enhancements

What is it?

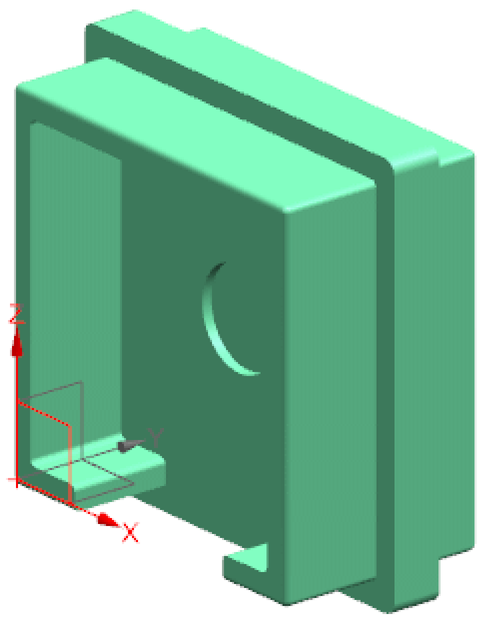
You can now control the size of the CSYS and Datum CSYS display and also control its visibility through the model. You can display the View Triad on the bottom right corner of the graphics window.

- To automatically update the size of the CSYS or Datum CSYS when you zoom in or zoom out, clear the **Fixed Size on Screen** check box.

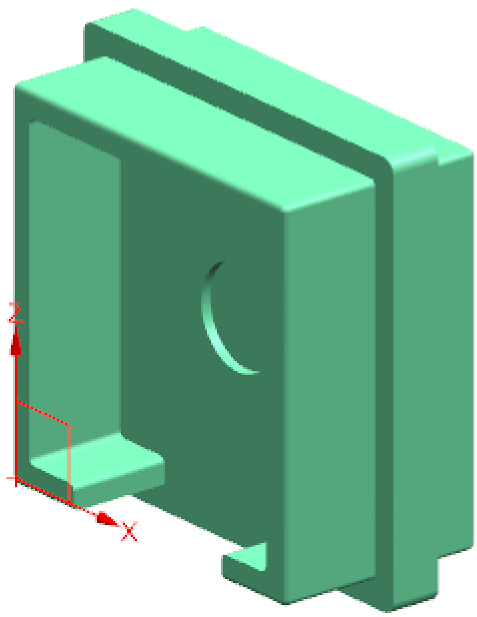


☑ **Fixed Size on Screen** ☐ **Fixed Size on Screen**

- To hide the visibility of the CSYS and the Datum CSYS through solid and non-transparent models, clear the **Show Through** check box.



☑ **Show Through**



☐ **Show Through**

- To display the View Triad on the bottom right corner of the graphics window, use the **Bottom Right** option.

Where do I find it?

Application	Gateway
Command Finder	Visualization Preferences
Location in dialog box	**Visualization Preferences** dialog box→**View/Screen** tab→**Part Settings** group

Rendering enhancements

What is it?

Scene Editor enhancements

You can now:

- Use non image-based advanced light effects such as highlights and shadows to obtain more realistic renderings.

  To do this, use the following options.

  - **Use with Advanced Studio Image-based Lighting**
  - **Use with Ray Traced Image-based Lighting**

  The **Generate High Quality Image Shadows** option on the **Shadows** tab is no longer available.

- Align the direction of the environment image with the floor plane direction.

  To do this, set the following.

  - **Up Vector** = **Align with Floor Plane**

- Use the same stage settings such as size, offset, and floor plane for different system scenes.

  NX retains the stage settings when you change the system scene.

System Scenes and System Materials enhancements

You can now use various industry specific visualization system scenes and system materials. The system scenes are organized under the new **Indoor**, **Outdoor**, and **Studio** categories.

English and **Metric** materials are no longer available.

Studio Task enhancements

You can now access the following commonly used commands using the **Studio Task** shortcut menu.

- **Select All Objects with this Material**
- **Edit Material**
- **Remove Material Assignment**
- **Apply Selected Material**
- **Inherit Material**

Where do I find it?

Scene Editor

Command Finder	**Scene Editor**
Location in dialog box	**Lights** tab→**Use with Advanced Studio Image-based Lighting** and **Use with Ray Traced Image-based Lighting** **Global Illumination** tab→**Up Vector** list→**Align with Floor Plane**

System Scenes and System Materials

Resource bar	**System Scenes** tab, **System Materials** tab
Command Finder	**System Scenes**, **System Materials**

Studio Task

Command Finder	**Studio Task**

Graphics window	Right-click the object→[command]

Displaying facets

What is it?

You can now:

- Control the amount of memory that NX uses for the display facet cache.

  To do this, use the **Facet Cache Memory** option. You can set the default value using the **Facet Cache Memory Level** customer default.

- Save display facets for advanced visualization views of solid bodies in the part file.

  To do this, use the **Save Advanced Display Facets** option. You can set the default value using the **Save Advanced Display Facets** customer default.

  Tip

  To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

Why should I use it?

Use these settings for faster rendering of solid bodies.

Where do I find it?

Command Finder	**Visualization Performance Preferences** 
Location in dialog box	**General Graphics** tab: **Session Settings** group→**Shaded Views** sub group →**Facet Cache Memory** **Part Settings** group→**Save Advanced Display Facets**

Roles enhancements

What is it?

The icon for the **Roles** tab has changed from  to .

The **System Defaults** category is now called **Content**. The icons for the roles are updated. A new **Layout** role in this category replaces the **Power Drafting Essentials** and **Power Drafting Advanced** roles. The **Layout** role provides a wider set of commands required for 2D conceptual design in the Drafting application.

An additional **Presentation** category is available. Select a role from this category that best suits your display device. The roles in this category do not change the commands that are displayed on the Ribbon bar. That content is specified by the role you select in the **Content** category. The available options in the **Presentation** category are:

- **Default**
- **Touch Panel**
- **Touch Tablet**

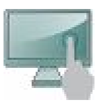
For more information related to these roles, see *Touchscreen gestures for NX*.

Where do I find it?

Resource bar	**Roles**  tab

Touchscreen gestures for NX

Use the **Touch Mode** command to work efficiently in NX on touchscreen devices. Select the NX presentation role for your device from the **Presentation** category in the **Roles** palette.

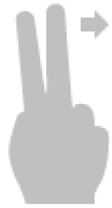
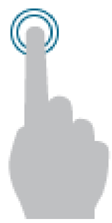
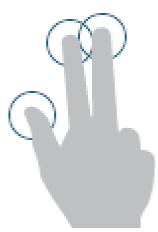
 **Touch Panel**	Optimizes the NX interface presentation to best match a touch panel monitor. This role displays the undocked Ribbon bar at the bottom of the interface and increases size and space around the interface elements such as Ribbon bar tabs, dialog box buttons, and so on.
 **Touch Tablet**	Optimizes the NX interface presentation to best match a small touch tablet. This role displays the Ribbon bar without text, removes the Border bars and the Title bar, and increases size and space around the interface elements such as Ribbon bar tabs, dialog box buttons, and so on.
 **Default**	Optimizes the NX interface presentation to best match a non-touch monitor. This role displays the default NX interface.

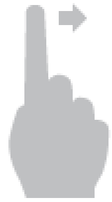
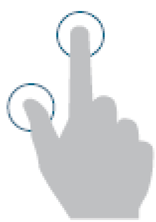
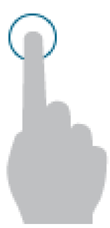
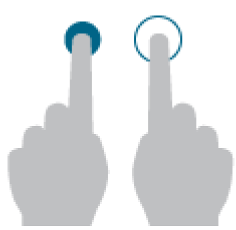
To manipulate views by panning, zooming, and so on, use touch gestures. To scroll, move on-screen handles, and so on, use a stylus or touch gestures.

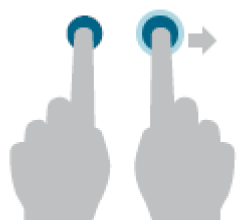
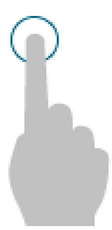
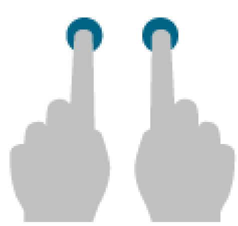
In addition to the simple tap gesture, there are two press gestures that you use for different actions.

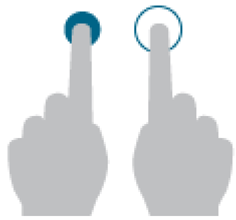
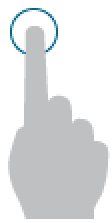
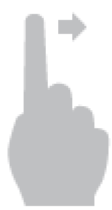
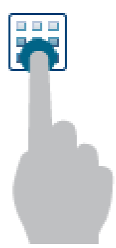
Short press	Touch the screen briefly as if you are pressing a keyboard button.
Long press	Touch the screen for about a second.

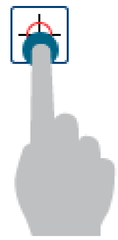
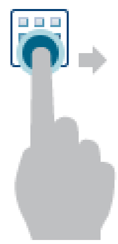
Action	Gesture	Description

Pan		Drag the object in the view with two fingers.
Rotate		Drag the object in the view with one finger. To rotate about an individual view axis, drag inside the view borders.
Fit		Double-tap anywhere in the view background.
Zoom		Touch the object with two fingers and move your fingers towards each other to zoom out. To zoom in, move your fingers away from each other.
Orient the view		Tap with three fingers anywhere in the view.
Rotate about a point		Short press at the location to rotate about and then move your finger in the direction in which you want to rotate the object.

Move an object or an on-screen handle		Drag the object with one finger.
View shortcut menu and shortcut toolbar		Long press and release anywhere in the view background.
View object-specific shortcut menu and shortcut toolbar		Long press the object and then release.
Select the next option in a dialog box or complete an action		Tap with two fingers anywhere in the view.
Select an object		Tap the object.
Deselect an object		Hold any location in the view with one finger, and then tap each object that you want to deselect.

Deselect all objects		Tap anywhere in the view background.
Select multiple objects using the lasso or rectangular selection		Long press, and then form a rectangle or a lasso around the objects that you want to select.
Deselect multiple objects using the lasso or rectangular selection		Hold any location in the view with one finger, and then with another finger, long press first and then form a rectangle or a lasso around the objects that you want to deselect.
View the QuickPick list for selection objects		Short press the desired object and then release.
Select an item in the QuickPick list		Tap the item and then either tap the **OK** button or tap two fingers anywhere in the view.
View the QuickPick list for deselection objects		Hold any location in the view with one finger, and then short press the desired object and release.

Browse to an item in the QuickPick list		Tap the item or tap anywhere in the view to advance to the next item.
Return to the previous item in the QuickPick list		Hold any location in the view with one finger, and then tap anywhere in the view with another finger.
Input numeric values in the graphics window		Tap the numeric input field to display **Numeric Keypad** and **Full Keyboard** . Tap the **Numeric Keypad** or **Full Keyboard** to display the respective touchpad and then tap the desired buttons.
Scroll ribbon tabs		Swipe the ribbon horizontally with one finger.
Select a view command on a radial toolbar		Tap the **Radial Spot** to display the radial toolbar with view commands, and then tap the desired command. You can also swipe from the **Radial Spot** in the direction of the desired command.

Select an object-specific command on a radial toolbar		Tap the **Object-specific radial spot** to display the radial toolbar with object specific commands, and then tap the desired command. You can also swipe from the **Object-specific radial spot** in the direction of the desired command.
Move the **Radial Spot**		Press and hold the **Radial Spot** with one finger until its appearance changes, and then drag it.

Where do I find it?

Prerequisite	Touchscreen
Command Finder	**Touch Mode**

Accessing image files from Teamcenter

What is it?

You can now import raster image files from Teamcenter to start your CAD geometry. You can also use image files stored in Teamcenter as decals on a face or body.

- To import a raster image file, use the **Raster Image** command.
- To use a stored image file as a decal, use the **Decal** command.

The image file is embedded in the NX part file and retains no associativity to the image file in Teamcenter.

Where do I find it?

Application	Gateway and Modeling
Prerequisite	Teamcenter Integration for NX
Command Finder	**Raster Image** **Decal**

Saving display settings in a bookmark for the displayed part

What is it?

Bookmarks are enhanced in the following ways.

- Bookmarks are now available in an HD3D tool.
- Bookmarks now capture many display settings of the current displayed part.

Applying a bookmark does not change the current application. For example, suppose you create a bookmark in the Modeling application and then switch to the Drafting application. If you open your bookmark, NX returns to the display state and displayed part captured by the bookmark but remains in the Drafting application.

Why should I use it?

Use bookmarks to quickly return to previous displayed states. For example, as you analyze your model, you can create a bookmark whenever you find a problem. Later, when you discuss your results with others, you can open each bookmark to show the problems that you found.

Where do I find it?

Resource Bar	**HD3D Tools→Bookmarks**

System attributes in NX

NX now provides system attributes related to material and mass properties and their templates.

You can use the customer defaults categorized under **Gateway→Materials/Mass→Attributes** tab to let NX set the system attributes. You can also set the title aliases such that they appear in the dialog boxes with the titles set in the previous releases. NX uses the same default strings for the title aliases as the pre-NX 10 default strings for the titles.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

NX 10 title for system attributes	NX 10 default for title aliases (pre-NX 10 default title)
NX_Density	MassPropDensity
NX_Mass	MassPropMass
NX_Weight	MassPropWeight
NX_Volume	MassPropVolume
NX_Area	MassPropArea
NX_Material	Material
NX_ObjectMaterial	ObjectMaterial
NX_MaterialMultipleAssigned	MaterialMultipleAssigned
NX_MaterialMissingAssignments	MaterialMissingAssignments

NX_MaterialPreferred	MaterialPreferred

Note

As system attributes may affect applications that use existing titles, you can use NX Open API to set the system attributes using the aliases set in the customer default.

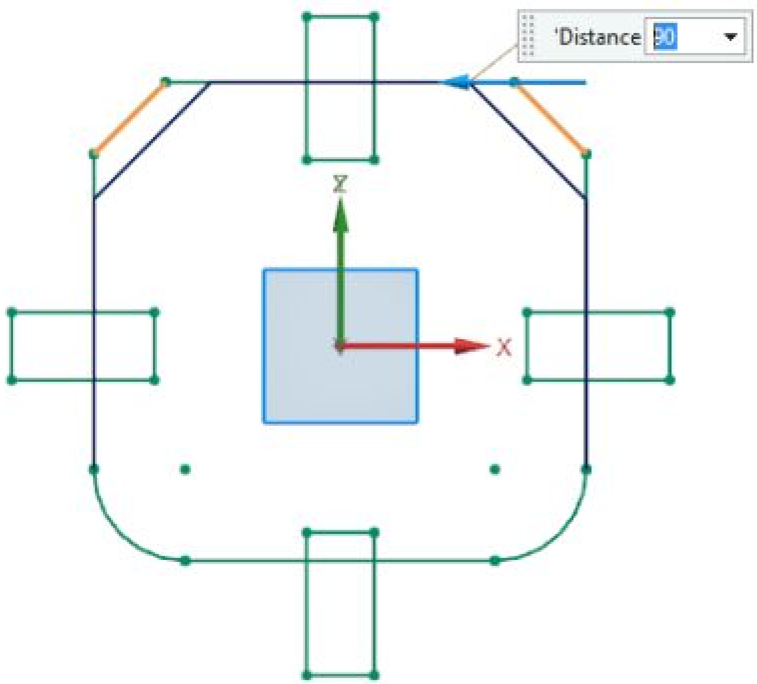
Chapter 3: CAD

Sketching

Resize Chamfer Curve

What is it?

Use this command to resize one or more chamfers in a sketch. You edit the chamfer using the customary chamfer offset distance, regardless of how the sketch was dimensioned and constrained. NX makes the edited chamfers symmetric using your specified offset distance.



Why should I use it?

You can edit sketch chamfers with or without constraints.

Where do I find it?

Application	Any application that uses Direct Sketch or the Sketch Task Environment
Prerequisite	You must be working in an active sketch.
Command Finder	**Resize Chamfer Curve**

Curve Finder options

What is it?

The **Curve Finder** options in the 2D Synchronous Technology **Move Curve** and **Offset Move Curve** commands now recognize the following geometric conditions.

- **Select Tangent**
- **Select Parallel**
- **Select Collinear**
- **Select Concentric**
- **Select Equal Radius**
- **Select Symmetric**
- **Select Offset**
- **Select Coincident Vertex**
- **Select Vertex on Curve**
- **Select Chamfer**

The settings **Select Offset**, **Select Coincident Vertex**, **Select Vertex on Curve**, and **Select Chamfer** are new in this release.

When you edit curves, the selected geometric conditions are maintained during the edit.

Why should I use it?

The support of these new geometric conditions is useful when you need to edit large sketches that do not have constraints.

Where do I find it?

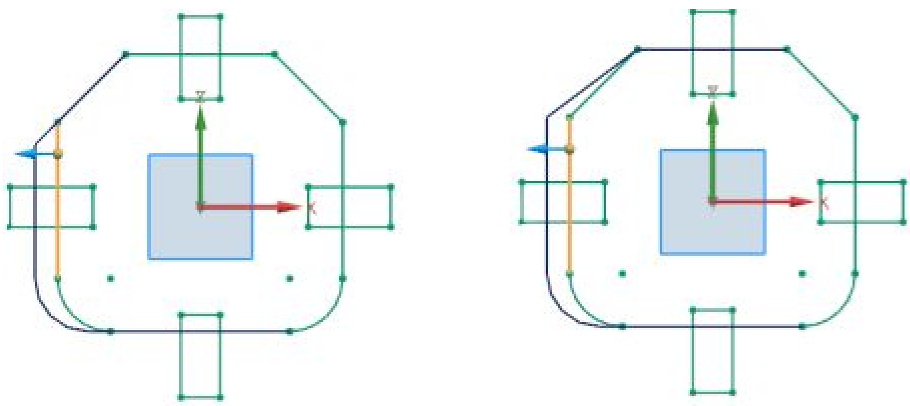
Application	Any application that uses Direct Sketch or the Sketch Task Environment
Prerequisite	You must be working in an active sketch.
Command Finder	**Move Curve** **Offset Move Curve**

2D Synchronous Technology settings

What is it?

In 2D Synchronous Modeling command dialog boxes, new options are added to the **Settings** group to control the angle of adjacent sides, and to control the behavior of arcs.

Size Lets you trim and extend the length of curves as you move them, or to maintain their original size and adjust adjacent curves.

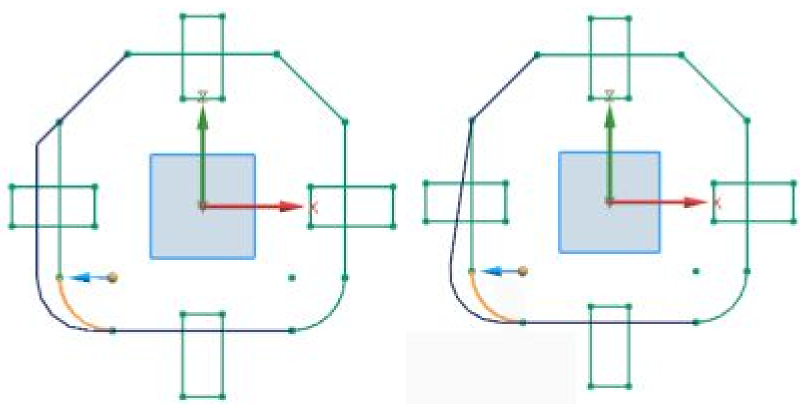


Size = **Trim and Extend** **Size** = **Maintain**

Keep Tangent Controls how NX maintains tangency of arcs when you edit curves.

Keep Orthogonal Keeps adjacent sides orthogonal when you move curves.

Clear this check box to allow NX to change the angle of adjacent sides.



Where do I find it?

Application	Any application that uses Direct Sketch or the Sketch Task Environment.
Prerequisite	You must be working in an active sketch.

Command Finder	**Move Curve** **Offset Move Curve** **Resize Curve**

Optimize 2D Curve

What is it?

Use **Optimize 2D Curve** to clean up and simplify imported 2D curve geometry and points, letting you work more efficiently with migrated 2D data.

You can specify **Distance Threshold** and **Angle Threshold** limit values to restrict which points and curves are optimized.

The optimize process:

- Cleans up curves and points
  - Removes overlapping curve chains (connected sets of curves that are part of other curve chains).
  - Removes small curve segments whose length is within the specified distance threshold.
  - Removes selected points if the **Include Point** option is active.
- Simplifies splines and conic curves
  - Replaces splines or conic curves with lines that meet the distance threshold.
  - Converts splines or conic curves to arcs that meet the distance threshold.
  - Refits splines to remove C0 knot points or self-intersections when needed.
  - Splits splines at their C0 knot points when they are 1 degree polylines.
- Connects and merges splines and lines
  - Removes gaps between lines that fall within the specified distance threshold.
  - Removes overlaps between lines that fall within the specified distance threshold.
  - Merges connected collinear lines.
  - Removes gaps between splines or conic curves and other curves if the gap is within the specified distance threshold.
  - Removes overlaps between splines or conic curves and other curves if the gap is within the specified distance threshold.

   - Removes the gap or overhead between the end of a curve and the middle of a curve (point on curve) if the gap or overhead is within the specified distance threshold.

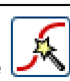
- Aligns arcs and circles
   - Merges connecting arcs and circles with the same radius within the specified distance threshold.
   - Recreates arcs that are tangent to two curves within the angle threshold with the same radius.
   - When two such arcs are tangent to each other, recreates the arc with the larger radius first using the same radius.
   - When an arc is tangent to one curve within the angle threshold, recreates the arc tangent to the curve with the same radius.
   - If a circle is concentric to an arc within distance threshold, it is made concentric to the arc.
- Updates downstream features that reference the selected curves after the optimization if they are outside a sketch.
- Creates a new sketch if the **Create Sketch** option is selected and there is no sketch currently active.

You can perform the optimization on a sketch before or after you import it.

Why should I use it?

Imported curve geometry that is not first optimized may be of insufficient quality for synchronous edit.

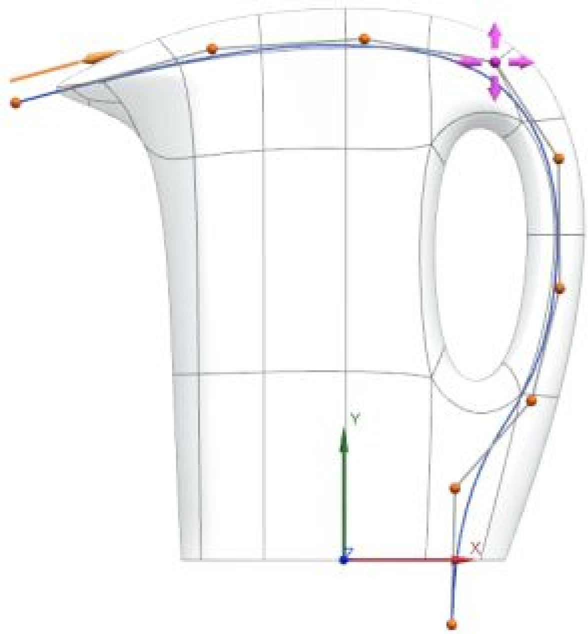
Where do I find it?

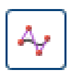
Application	Modeling
Command Finder	Optimize 2D Curve 

Creating sketch splines

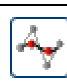
What is it?

NX is enhanced to allow more control over spline poles when you create or edit splines in a sketch.



Studio Spline command	When you create or edit splines by poles, you can infer G1 and G2 Tangent Continuity constraint at the start and end poles. You can drag the defining poles of the spline to create constraints.
Top Border bar	A new **Poles**  snap option is added to the Top Border bar to allow snapping to poles. For example, use this option when creating dimensional constraints to the pole locations.
Point construction	In commands that allow you to create or select a point, a new **Spline Pole**  option is available in the point list. Use this option to select spline poles.

Where do I find it?

Application	Modeling
Prerequisite	You must be working in an active sketch.
Command Finder	**Studio Spline** 

Trimming sketch recipe curves

What is it?

When you use the commands **Quick Trim**, **Make Corner**, **Fillet**, and **Chamfer**, you can trim recipe curves without breaking the associativity from their parent geometry, if the **Create Inferred Constraints** option is turned on. If a recipe curve is extended using one of these commands, an alert message warns you that the associativity will be removed.

Recipe curves are curves that are associatively projected to the sketch using either the **Project Curve** or **Intersection Curve** command.

Where do I find it?

Application	Modeling
Prerequisite	You must be working in an active sketch. The **Home** tab→**Direct Sketch** group→**Create Inferred Constraints** option must be turned on.
Command Finder	**Quick Trim** **Make Corner** **Fillet** **Chamfer**

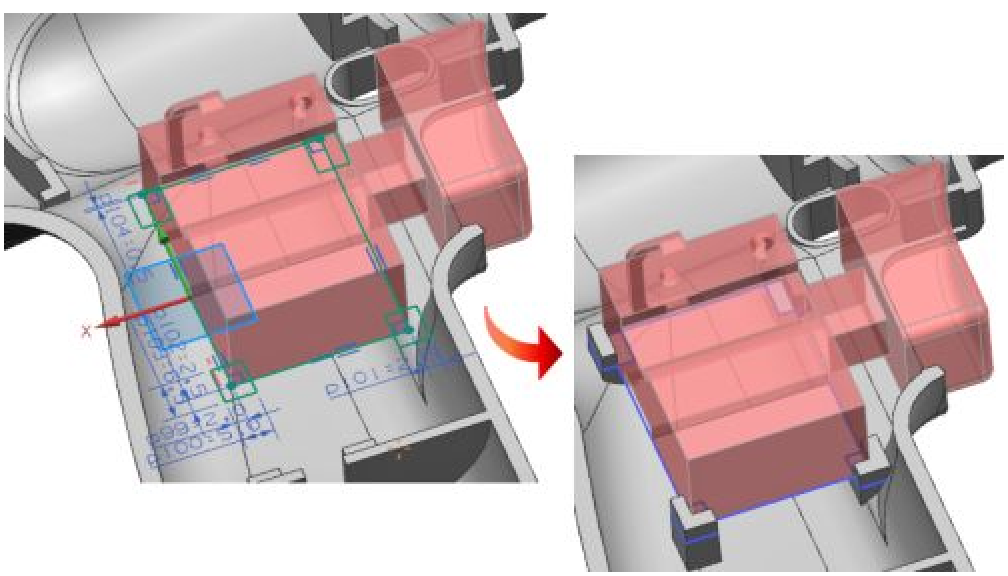
Creating WAVE links in a sketch

What is it?

You can create links to geometry outside the work part, using constraint and dimension Sketch commands.

- To select geometry outside the work part, set the **Selection Scope** option to **Entire Assembly**.
- To create a WAVE link to the selected geometry, turn on the option **Create Interpart Link**. If a non-associative link is desired, turn off the option **Create Interpart Link**.

In this example, a sketch is used to create pads in the handle of a cordless drill to support the trigger assembly. With the handle as the work part, the sketch is constrained to geometry in the trigger assembly part.



Why should I use it?

These options allow you to create associative or non-associative links from a sketch to geometry in another assembly component.

Where do I find it?

The options **Selection Scope** and **Create Interpart Link** are located in the Top Border bar when you create sketch dimensions or geometric constraints.

Application	Modeling
Prerequisite	You must be working in an active sketch in an assembly.
Command Finder	**Rapid Dimension** **Geometric Constraints**

Part Module

Part Module enhancements in the Part Navigator

What is it?

In the **Part Navigator**, you can now:

- Right-click a part module and choose **Activate, Show Only, and Expand**. This new shortcut activates the selected part module, shows only objects in the part module, and expands the part module features to make the part module easier to work on. You can also double-click on a part module to invoke this command.

- Drag features from one part module to another part module to reorder or reorganize the structure in a part module.

- Drag features into an existing part module to add structure to a part.

- Delete a part module without deleting its features to remove structure from a part.

- Drag a part module into another part module to nest them. You can also drag a nested part module out of part module to un-nest the part modules. This functionality replaces the **Nest Part module** and **Un-nest Part Module** commands.

- Update out of date linked part modules when you click the out-of-date icon in the **Up to Date** column. When you click the out-of-date icon for a feature, NX updates that feature and any out-of-date parents. When you click the out-of-date icon for a part module, NX updates that part module, all its member features, and any out of date parents.

Save linked part module enhancement

What is it?

Linked part module files can no longer be saved in an active state. When you save a linked part module part, NX determines if the linked part module is active. If the linked part module is active, NX will automatically:

1. Deactivate the linked part module.

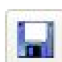
2. Save the linked part module file.

3. Activate the linked part module.

   Note

   When NX deactivates the linked part module, the module may update if it has pending changes. For a complex linked part module, this could take some time, and all the normal ramifications of an update will be encountered. Beyond the linked part module itself, normal NX interpart update behavior will apply.

Where do I find it?

Application	Modeling, Sheet Metal Design, and Shape Studio
Command Finder	**Save** 

Update Input References and Update Output References enhancements

What is it?

The **Update Input References** and **Update Output References** commands check if the latest version of the parent is loaded into memory.

While working on linked part module parts, the **Update Input References** command now does the following:

- If the main part is closed, NX will display the following message which gives you the opportunity to open a read only version of it:

  ```
  The operation attempted cannot be executed because the corresponding Linked Part Module file is not loaded in the session.

  Do you want to load the corresponding Linked Part Module file?
  ```

- If the main part is open but it's not the latest version on disk, NX displays the following message which can open a read only version of it:

  ```
  The command will load the latest disk version of the main and/or associated linked part module parts.  Unsaved changes in locally loaded parts will be lost.

  Do you want to continue with the operation?
  ```

While working on main parts, the **Update Output References** command now does the following:

- If the linked part module part is closed, NX displays the following message which gives you the opportunity to open a read only version of it:

  `The operation attempted cannot be executed because the corresponding Linked Part Module file is not loaded in the session.`

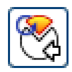
  `Do you want to load the corresponding Linked Part Module file?`

- If the linked part module part is open but it's not the latest version on disk, NX displays the following message which can open a read only version of it:

  `The command will load the latest disk version of the main and/or associated linked part module parts.  Unsaved changes in locally loaded parts will be lost.`

  `Do you want to continue with the operation?`

Where do I find it?

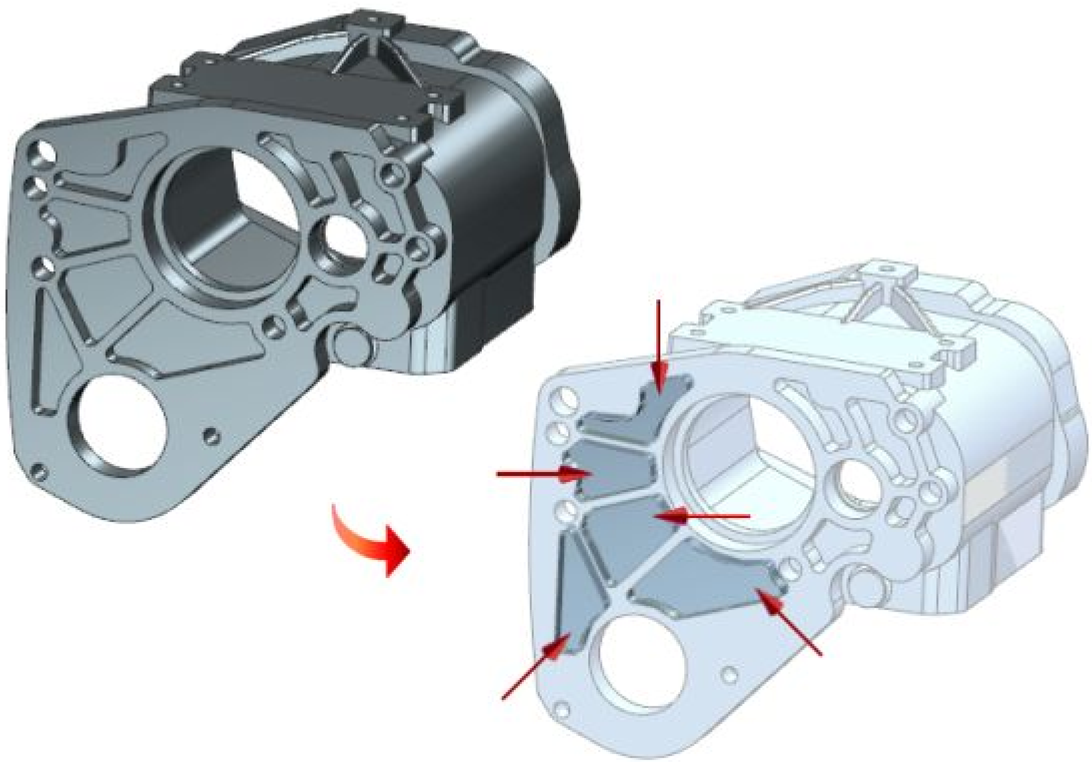
Application	Modeling, Sheet Metal Design, and Shape Studio
Command Finder	**Update Input References**  **Update Output References** 

Shared Body enhancement

What is it?

When you activate a part module that contains a **Shared Body**, all the modifiable objects are emphasized. This makes it easier to locate shared bodies in the graphics window.

In this example, the five pockets are modifiable objects in a part module.

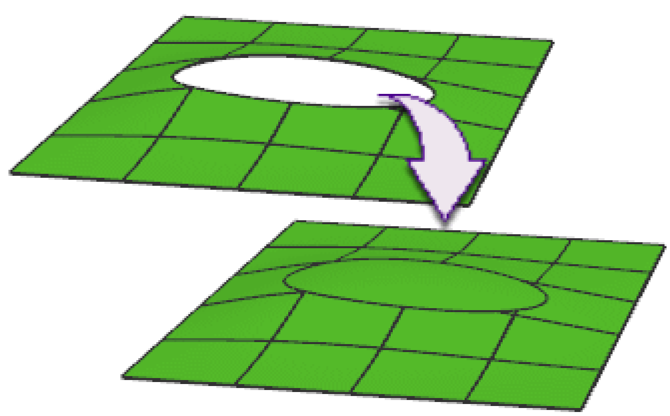


Modeling

Fill Surface enhancements

What is it?

The **Fill Surface** command has three changes to the **Method** list in the **Shape Control** group of the dialog box.



- **Pass Through Curves** is now named **Fit to Curves**.
- **Fit to Facet Body** is a new shape control method. The fill is fitted to an existing facet body.

- Select of a set of facet body edges that are used to shape the surface.
- Specify the **Area of Control** and the **Attraction**.
- See the reports of the **Average** and **Maximum** errors.

You can set the **Default Edge Continuity**.

Why should I use it?

Use **Fill surface** to fill gaps or holes in any surface.

Where do I find it?

Application	Modeling
Command Finder	**Fill Surface**

Section Surface enhancements

What is it?

A **Body Type** option is added to the **Settings** group of the dialog box. The options are:

- **Solid**
- **Sheet**

The choice is made at creation or when editing the section body.

Why should I use it?

There is no longer a need to use the **Modeling Preferences** dialog box to make this choice.

Where do I find it?

Application	Modeling and Shape Studio
Command Finder	**Section Surface**

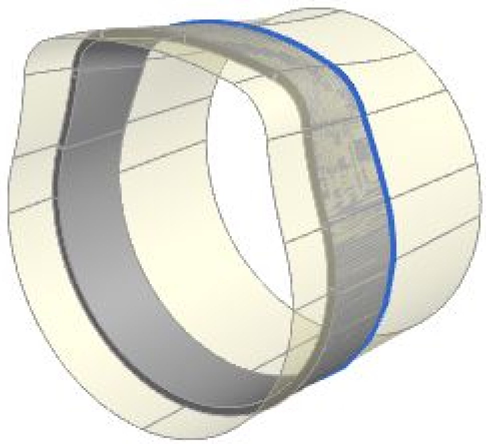
Aero Design

What is it?

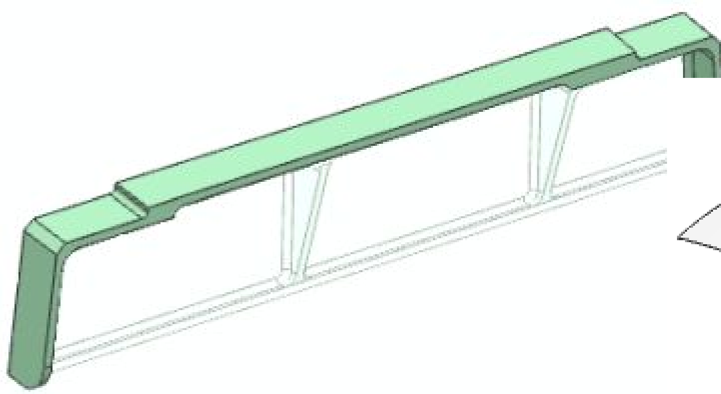
Aero Design is a group of four new commands specifically designed to help you create vehicular support structures but the commands are not limited to those applications.

Aero Flange

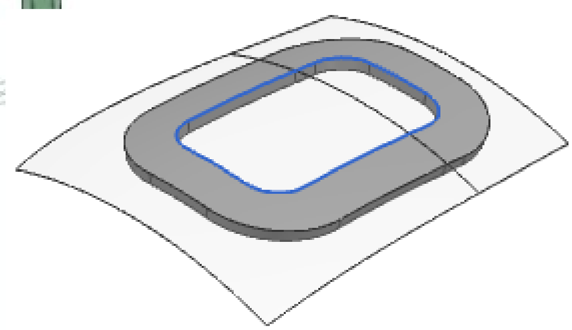
Use the **Aero Flange** command to create a flange body by offsetting from a specified region of a skin which forms the exterior of a vehicle or vessel.



To create flanges, both open and closed, you can specify parameters such as width, thickness, and offset.



Open flange as part of web

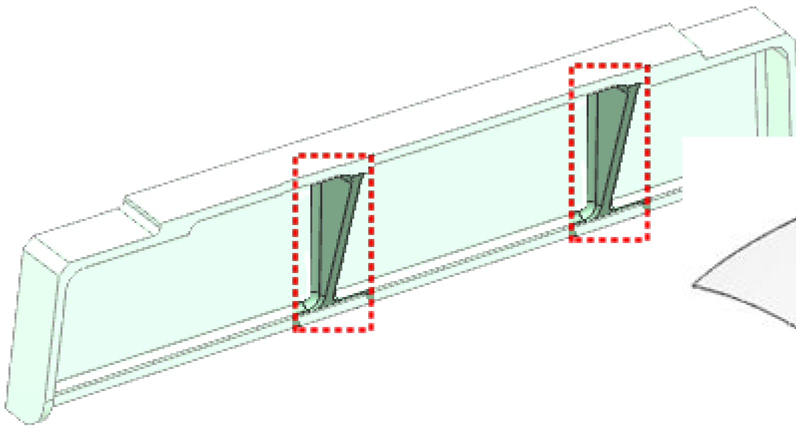


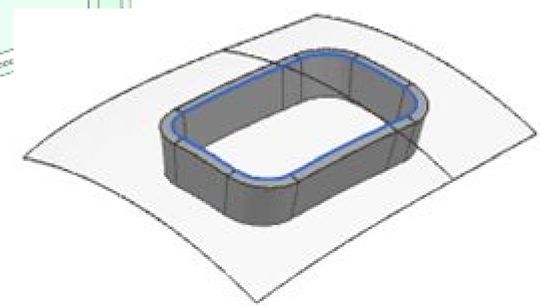
Closed flange as part of access panel

 Aero Rib

Use the **Aero Rib** command to create a thin-wall body by extruding from a face chain using a guide curve. You can create both open and closed ribs. You specify parameters such as height, length (non-closed ribs), thickness, and offset.

When referencing a planar skin, the rib body is created by extruding. For general cases, the rib body is generated by extending the guide curve from skin.



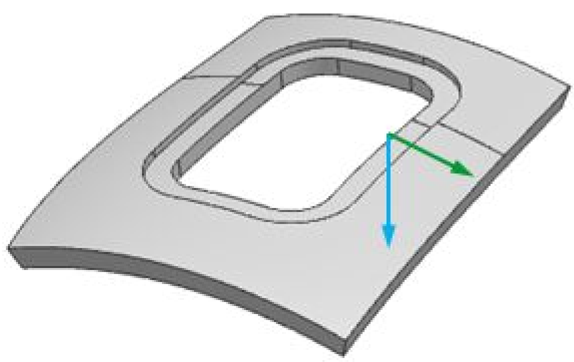


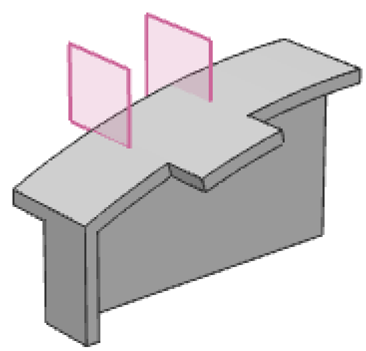
Aero ribs as part of web

Closed aero rib as part of web

Aero Shelf

Use the **Aero Shelf** command to offset faces that neighbor an edge to form a shelf along the edge of a support structure.



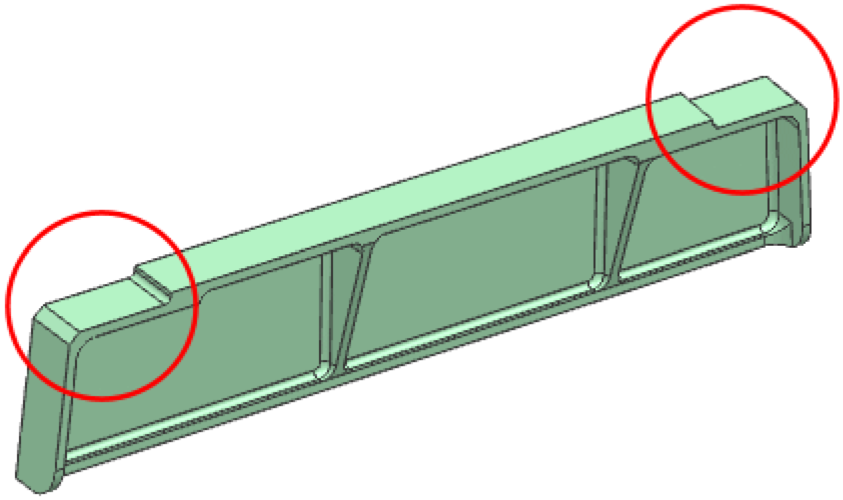


Closed aero shelf

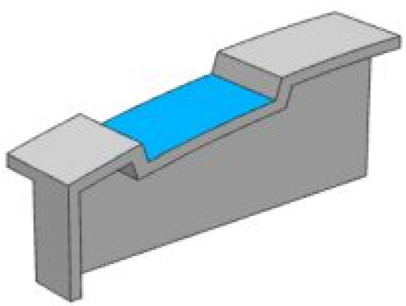
Aero shelf extension (two reference objects)

Aero Step

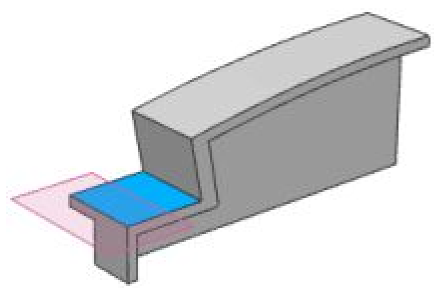
Use the **Aero Step** command to create a step feature that offsets a face region to add clearance where the part would mate with another part.



To create steps, you can specify parameters such as length, depth, thickness, and offset.



Aero step middle offset



Aero step offset to plane

Where do I find it?

Application	Modeling
Command Finder	**Aero Flange**, **Aero Rib**, **Aero Shelf**, **Aero Step**

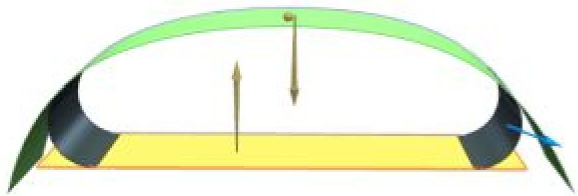
Face Blend enhancements

What is it?

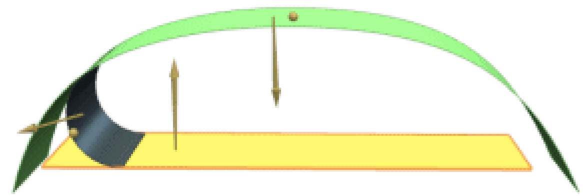
The **Face Blend** command is enhanced in the following ways:

- **Multiple blend solutions**

  You now have more choices in how multiple-solution face blends are created when specifying Rolling Ball, Naturally Varying, Circular, or Constant Radius blends that do not have any constraining or limiting geometry.



- In multiple-solution situations, all possible face blends that meet selection criteria are initially generated.
- In this example, both face blends yield one **Face Blend** feature.



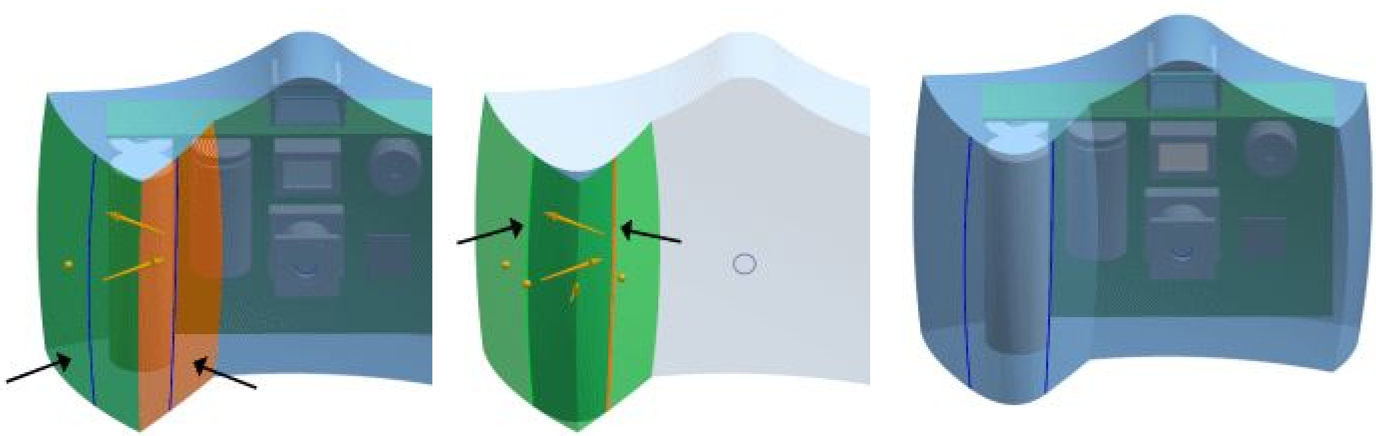
- You can bypass the default behavior by indicating which blend to keep by specifying a location.
- In this example, the blend is created as a non-associative **Body** feature.

- **Blend Face continuity**

  You can now create face blends that are either tangent continuous or curvature continuous to all face chains that define the face blend.

- **Blend constraint curves**

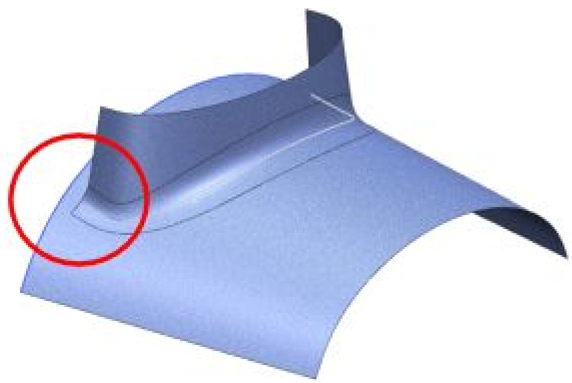
  You can now limit the extent of the face blend by selecting constraint curves that act as *hold lines*, one on each defining face.

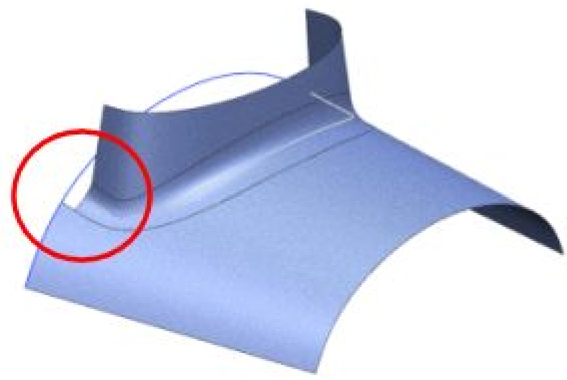


Face chains selected | **Constraint curves selected** | **Face blend to constraint curves**

- **New Trimming option**

  When the **Blend Faces** option is set to **Trim to Short Input Faces**, you can optionally extend the blend rail or hold line to the edges of the face chain on which it resides, and trim the long face chain to the extended rail or hold line.





Trim Input Faces to Blend Faces (checked)
Trim Long Input Faces to Extended Rail (unchecked)

Trim Input Faces to Blend Faces (checked)
Trim Long Input Faces to Extended Rail (checked)

- **Option name changes**

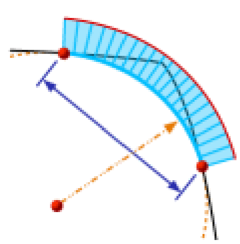
  The following **Shape** options in the **Cross Section** group are renamed from previous releases:

  Tangent Symmetric replaces **Symmetric Conic**.

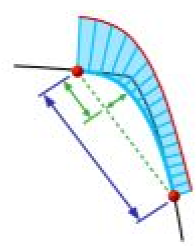
  Tangent Asymmetric replaces **Asymmetric Conic**.

- **Forced constant width method**

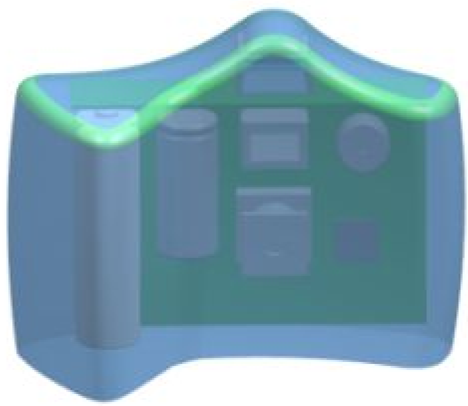
  You can now create face blends that have a constant rail-to-rail width. You can create both symmetric and asymmetric forced constant-width blends.



Symmetric



AsymmetricSkew ratio determines asymmetric positioning



Example of symmetric constant-width face blend

Note

These enhancements to **Face Blend** replace the **Soft Blend** command of previous releases.

To exactly mimic the behavior of **Soft Blend**, use the **Face Blend** options **Swept Section** with the **Two Constraining Curves** blend method.

Why should I use it?

- To determine how face blends are created when there are multiple possible solutions.
- To specify the type of continuity the blend has to its defining face chains.
- To define the blend using constraint curves.
- To trim the long face chain of a face blend to an extended rail or hold line.
- To create a symmetric or asymmetric constant-width blend.

Where do I find it?

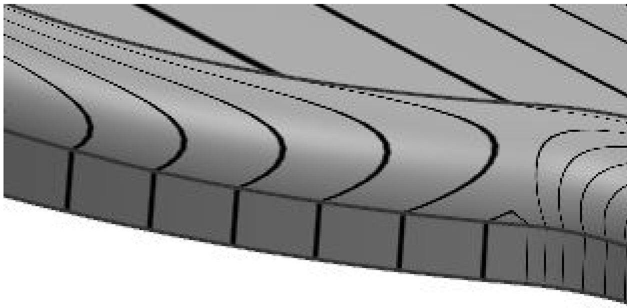
Application	Modeling
Command Finder	**Face Blend** 

Edge Blend enhancements

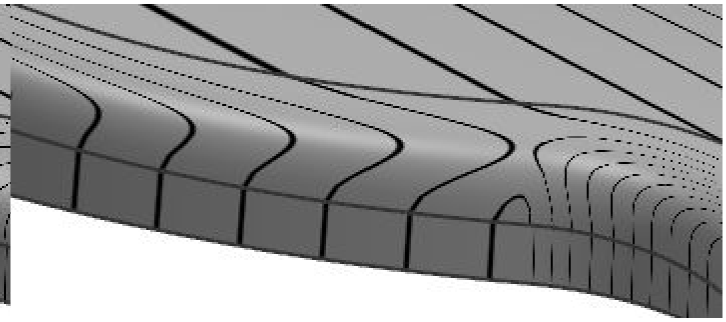
The **Edge Blend** command is enhanced in the following ways:

- **Curvature continuity**

  You can now create edge blends that are curvature continuous (G2) with their adjacent faces.



Blend Face Continuity = Tangent



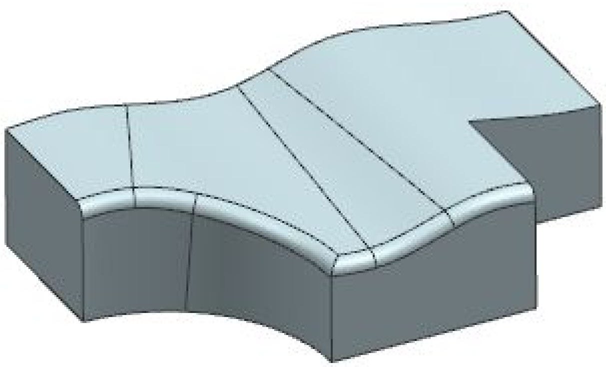
Blend Face Continuity = Curvature

Note

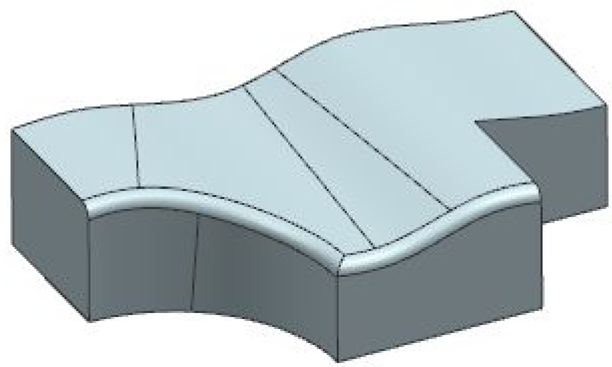
Curvature continuity is typically used by class A surface modelers to attain ideal aesthetic reflection quality.

- **Segment blending**

  You can now segment a blend face to match the segmentation of edge adjacent faces.



☑ **Segment Blend to Match Face Segments**



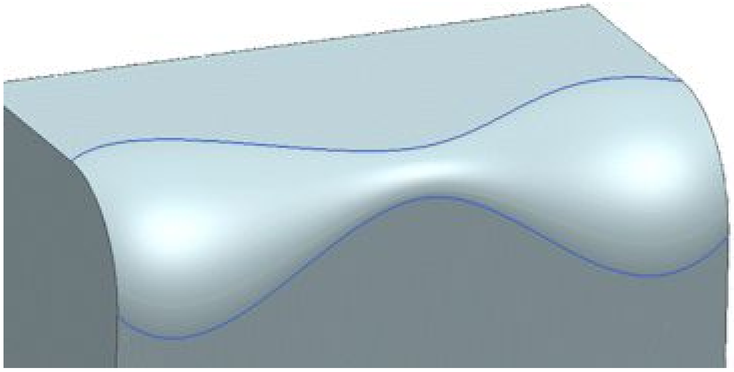
☐ **Segment Blend to Match Face Segments**

Where do I find it?

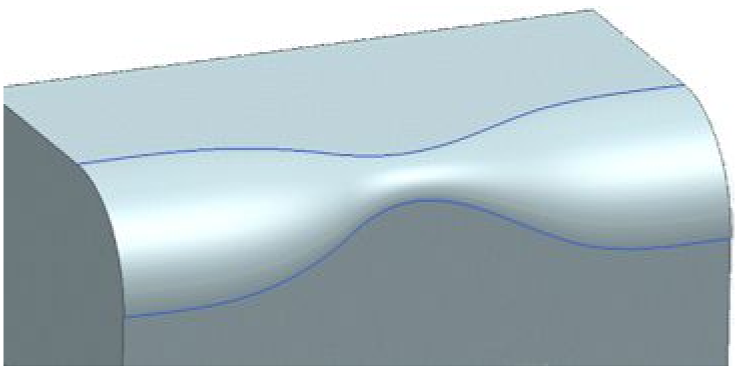
Application	Modeling
Command Finder	**Edge Blend**

Edge Blend enhancement (10.0.1)

When creating an **Edge Blend** with variable radius points, you can now specify that the radius function has zero slope at each end of the blended edge chain.



Zero Slope Radius Function at Chain Ends



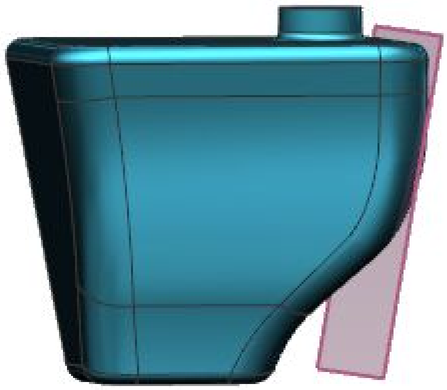
Zero Slope Radius Function at Chain Ends

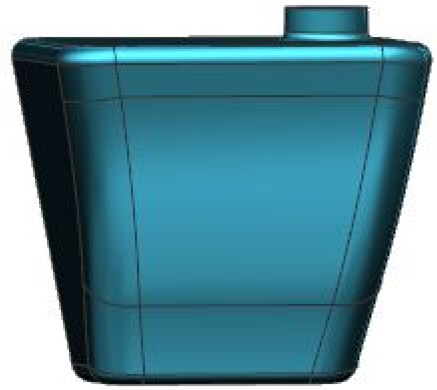
Where do I find it?

Application	Modeling
Command Finder	**Edge Blend**
Location in dialog box	**Settings→Zero Slope Radius Function at Chain Ends**

Replace Face enhancement

The **Replace Face** command is enhanced so that you can now select a datum plane as the Replacement Face.





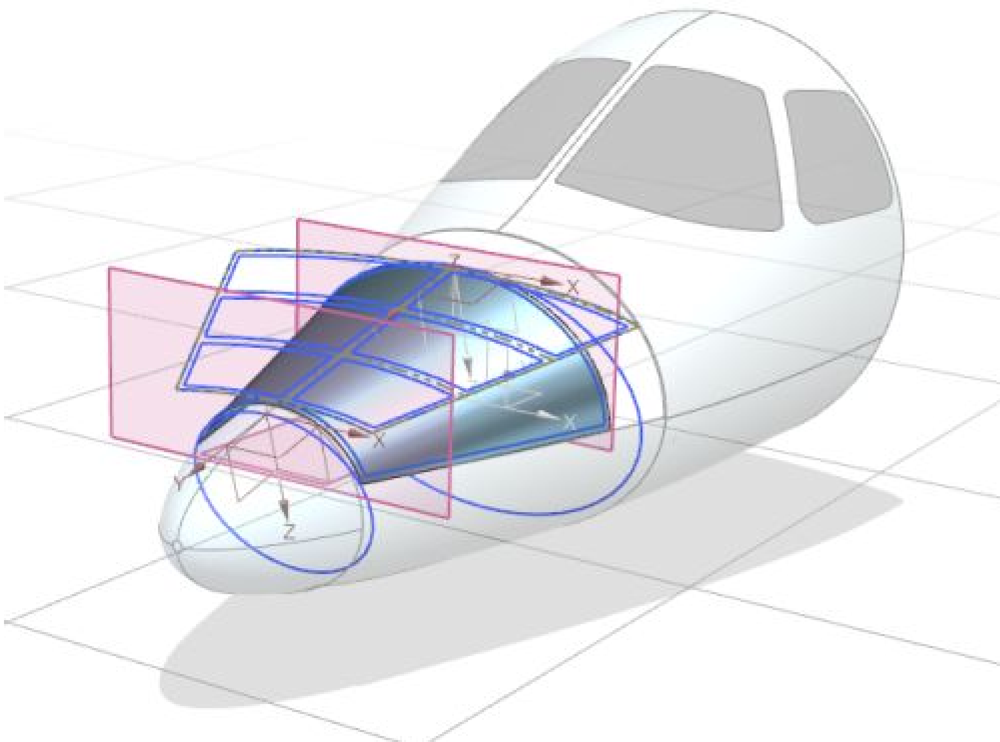
Where do I find it?

Application	Modeling
Command Finder	**Replace Face** 

Wrap/Unwrap Curve enhancements

What is it?

The **Wrap/Unwrap Curve** command now supports the wrapping and unwrapping of points and multiple ruled developable surfaces.



For **Wrap Curve**, the inputs are:

- Any set of curves, including points.
- Any set of developable surfaces onto which the wrapping is wanted.
- A plane tangent to any of the selected developable surfaces.

For **Unwrap Curve**, the inputs are:

- Any set of curves, including points, which lies on the given set of developable surfaces.
- Any set of developable surfaces from which the unwrapping is desired.
- A plane tangent to any of the those selected developable surfaces.

Why should I use it?

You can wrap and unwrap points from a cylindrical, conical, or ruled developable surface from or to a plane. You can also wrap and unwrap selected sets of curves lying on multiple ruled developed surfaces.

Where do I find it?

Application	Modeling
Command Finder	**Wrap/Unwrap Curve** 

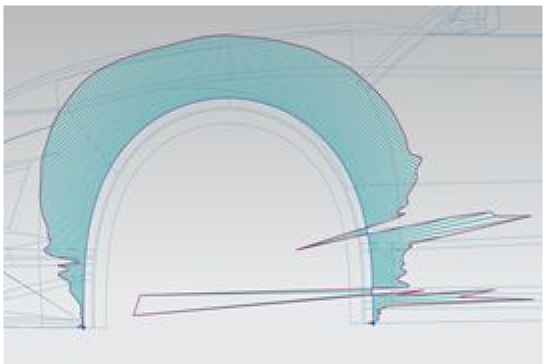
Offset Curve in Face enhancements

What is it?

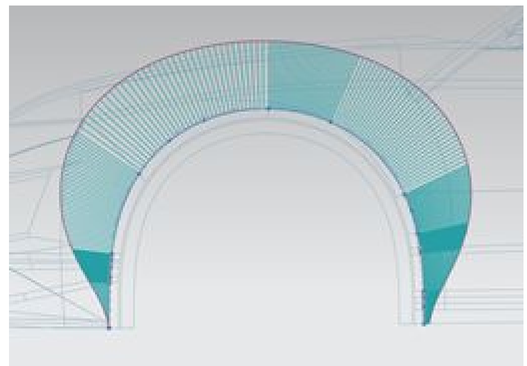
Offset Curve in Face has the following improvements:

- A new **Split Curve at Face Boundary** option to let you segment the output feature curve at each face boundary it crosses.
- Better curve fitting with **Advanced Curve Fit** methods.

These enhancements bring improved performance, accuracy, and new cross-face continuity and case handling capabilities to **Offset Curve in Face** feature curves, making them of the same high quality as that of other derived curves.



Offset Curve in Face curvature result previous to NX 10



NX 10 Result

Where do I find it?

Application	Modeling
Command Finder	**Offset Curve in Face**

Preview a failed or missing face during feature edit

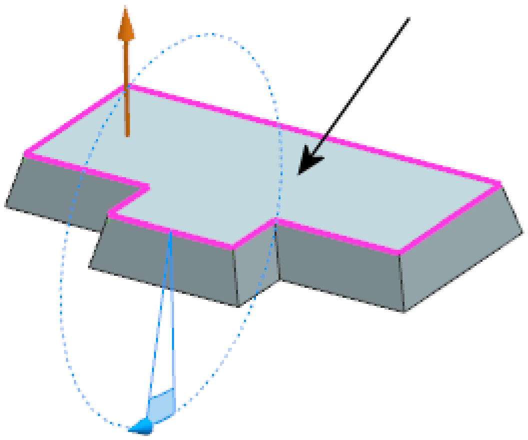
What is it?

If a missing object error appears when editing a body or a face, a preview can now show where in the model the missing object was located. You can view the preview using the **Show Previous State** option in the dialog box of the feature you are editing.

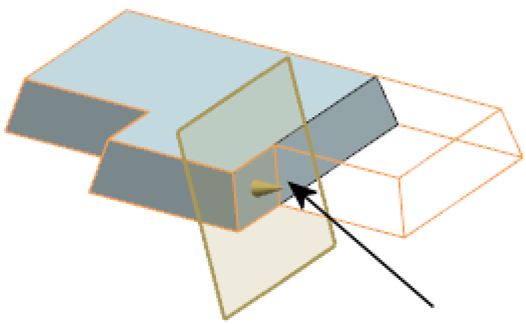
The missing object error could be due to any of the following:

- A previously existing face was consumed in a subsequent operation.
- The shape of a face or body was changed.
- A face or body was reordered to a different position in the **Part Navigator**.

The following sequence shows an example of how a previous state preview can be viewed during a feature edit:



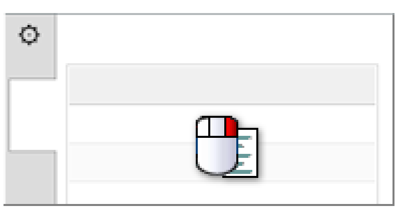
1. A 15° **Draft** feature is added to an extruded solid body.



2. The body is trimmed with a **Trim Body** feature.

   In the **Part Navigator** the **Trim Body** feature becomes the current feature.

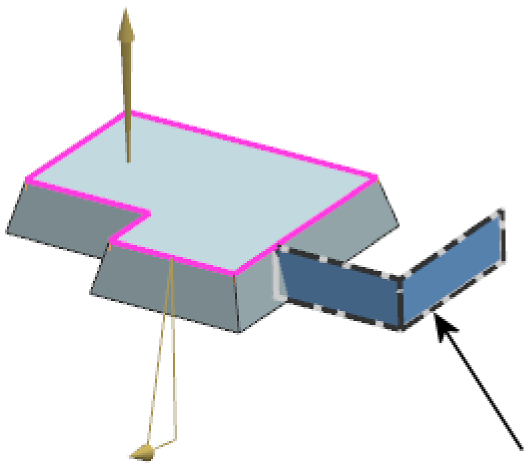
   - Model History
     - Datum Coordinate System (0)
     - Extrude (1)
     - Draft (2)
     - **Trim Body (3)**



3. In the **Part Navigator** the **Trim Body** feature is reordered before the **Draft** feature.

   The **Draft** feature becomes the current feature and its icon displays an alert indicating faces have been consumed.

   - Model History
     - Datum Coordinate System (0)
     - Extrude (1)
     - Trim Body (2)
     - **Draft (3)**



4. The **Draft** feature is edited and the **Show Previous State** option in the **Draft** dialog box is selected.

   The previous state preview of the consumed faces appears.

This new capability is similar to the existing behavior in **Edit Parameters**, where a missing edge error displays the preview of an edge that was consumed or missing from an earlier operation.

To support this enhancement the old **Save Data For Fast Rollback on Edit** customer default has been replaced by the new **Save Data for Feature Edit** customer default, with the following option choices:

- **None**
- **Fast Rollback**
- **Fast Rollback and Previous State of Failed Feature**

Use this new customer default to specify what additional data will be saved in the part to enhance feature edit.

Features that currently support Previous State Preview

Face Blend	**Offset Face**
Trim Body	**Draft**
Trim and Extend	**Delete Face**
Trim Sheet	**Draft Body**

Where do I find it?

Command Finder	**Customer Defaults**
Menu	**File**→ **Utilities**→ **Customer Defaults**
Location in dialog box	To activate: **Gateway**→ **General**→ **Part** tab→ **Save Data for Feature Edit** To use during edit: **Show Previous State**

Rollback data to memory to improve Modeling performance

What is it?

Use the **Load Rollback Data to Memory** customer default to allow NX to rollback its available data to memory instead of to the hard disk, thus improving the overall performance of Modeling operations.

The memory requirements of NX will rise accordingly while this customer default is enabled, and it is not recommended for systems with insufficient installed physical memory.

The amount of data that is rolled back is controlled by the current hardware configuration.

Why should I use it?

Most Rollback data will remain in memory while this customer default is enabled, reducing the time it takes to read and write to disk and increasing the speed of Modeling.

Where do I find it?

Customer Defaults	**Load Rollback Data to Memory**
Menu	**File**→ **Utilities**→ **Customer Defaults**→ **Modeling** → **General**→ **Update** tab→ **Load Rollback Data to Memory**

Offset 3D Curve enhancements

What is it?

The former **3D Curve Offset** command has been enhanced and renamed to **Offset 3D Curve**, and has been moved to the **Derived Curve** Group within the **Modeling** application.

The enhancements include:

- An **Associative** option to create associative **Offset 3D Curve** features.
- **Advanced Curve Fit** methods and options:
  - **Degree and Segments** method to specify the output curves using values for the degree and the number of segments.
  - **Degree and Tolerance** method to specify the output curves using values for the degree and tolerance.
  - **Keep Parameterization** method to inherit the parameters for degree, number of segments, and the pole and knot structure from the input curve (or in the case of an Intersection Curve, the input surface) and apply them to the output curve.
  - **Auto Fit** method to specify the minimum and maximum degree, the maximum number of segments, and the tolerance of the output curve.
- A **Join Curves** option to either maintain individual B-splines in the output curve, or join them in a single B-spline.
- A dynamic **Preview** mode available during both feature creation and edit.

These improvements bring **Offset 3D Curve** to the same level of capability and completeness as that of the **Offset Curve** command.

Where do I find it?

Application	Modeling
Command Finder	**Offset 3D Curve**

Enhancements to patterns with extracted bodies

What is it?

The **Pattern Feature** and **Pattern Geometry** commands now copy both positive and negative extracted symbolic threads and threads on positive holes for downstream operations.

- When extracting **Mirrored Body** and **Body** types, if the **Copy Threads** check box is selected, the positive or negative disposition of the extracted symbolic threads is maintained in subsequent patterning of those threads.
- This includes **WAVE** linked extracted geometry.
- This enhances **Pattern Feature** so that you can now copy extracted hole threads using either the **Variational** or the **Simple** pattern method. Previously only **Variational** was available for copying extracted threads.

Why should I use it?

To ensure correct patterning and mirroring behavior when extracting geometry and patterning it.

Where do I find it?

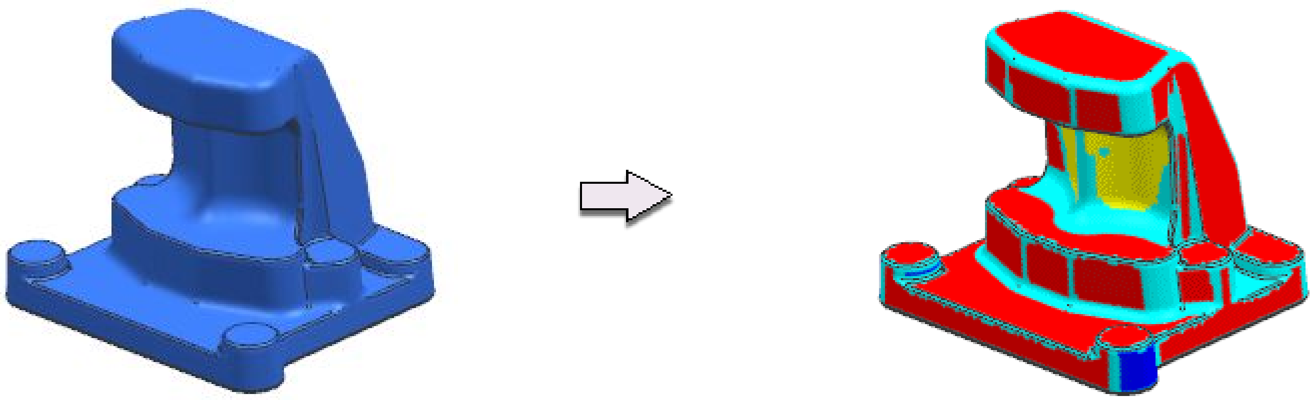
Application	Modeling
Command Finder	**Extract Geometry**, **Pattern Feature**, **Pattern Geometry**
Location in dialog box	**Extract Geometry** dialog box: **Type** list→**Mirror Body**, **Body** options→**Copy Threads** **Pattern Feature** dialog box: **Pattern Method** group, **Method** **Pattern Geometry** dialog box: **Settings** group→**Copy Threads**

Detect Primitives

What is it?

Use the **Detect Primitives** command to identify the shape types of a facet body and add color attributes to them.

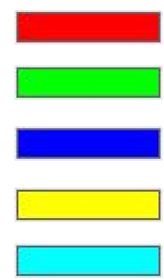
All color information is stored as Face Analysis data.



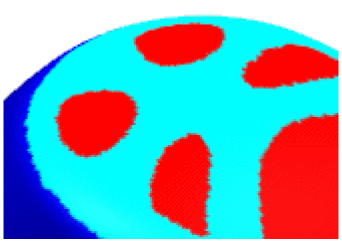
Each shape type is assigned a unique color attribute that is added to the vertices of the selected facet body.

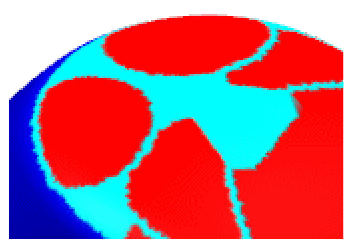
- To display the shape colors select a facet body.
- To disable the colors change the **Rendering Style**.
- To re-enable the colors set the **Rendering Style** to **Face Analysis**.
- To delete the color attributes select the facet body with the **Edit Object Display** command and clear the **Face Analysis** option.

Shape type classification



The **Curvature Sensitivity** option can narrow or broaden the transition range between planar shapes and other types.





Curvature Sensitivity low | **Curvature Sensitivity** high

The boundary between shape types appears in the **Other** shape color to help separate the main features.

You can configure any of the assigned colors by clicking the respective shape type in the dialog box.

Why should I use it?

You can create specific color attributes for facet bodies and use them for color coded selection using the **Fit Surface** command.

Where do I find it?

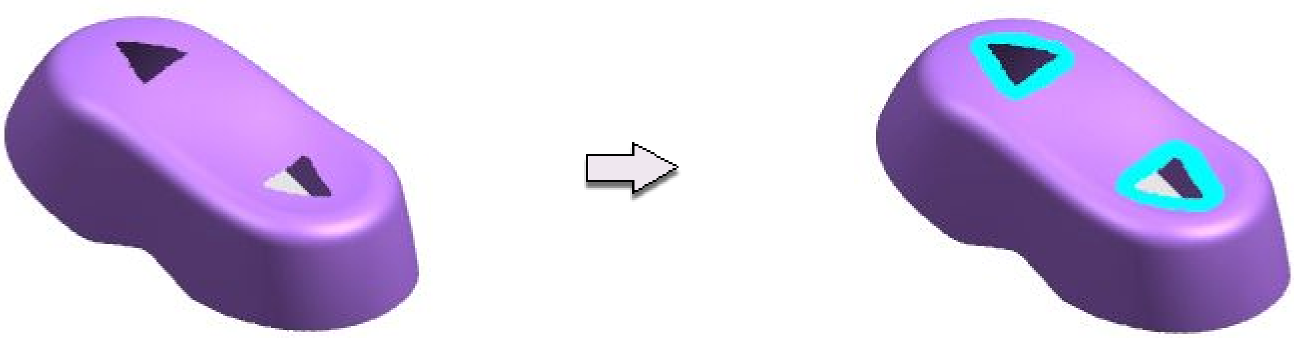
Application	Modeling and Shape Studio
Command Finder	**Detect Primitives** 

Paint Facet Body

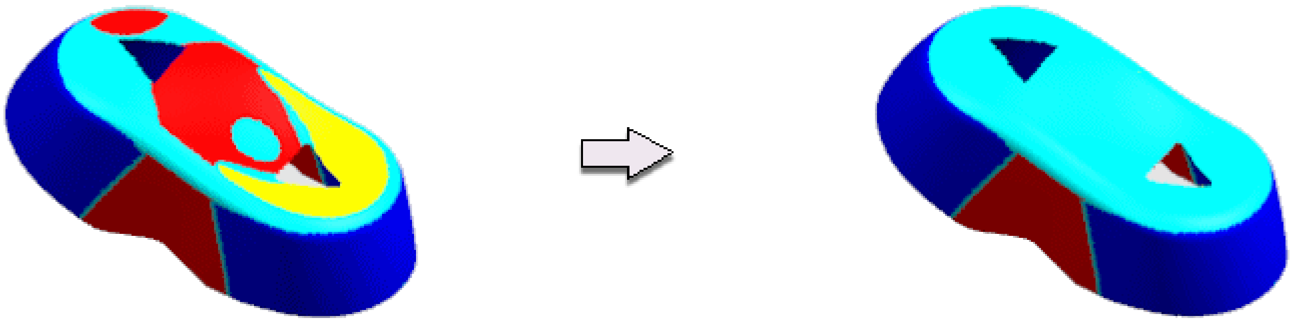
What is it?

Use the **Paint Facet Body** command to apply color attributes to facet bodies interactively using a digital paint brush.

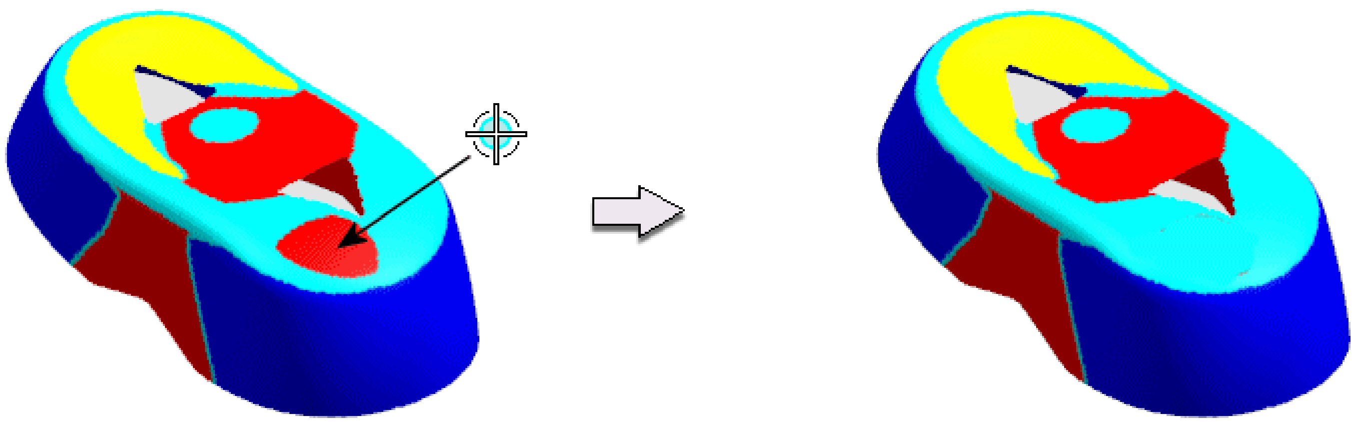
You can paint regions on facet bodies to call out areas of interest.



You can paint facet bodies to clean up color attributes that were automatically assigned by the **Detect Primitives** command, so the facet model is better suited for color coded selection by other tools.



If you only need to correct small irregularities from **Detect Primitives** output (as shown here), the **Paint Facet Body** digital paint brush lets you easily do so.



- You can specify your own paint colors or inherit them from a facet body.
- You can specify the size of the digital paint brush.

All color information is stored as Face Analysis data and can be deleted by selecting the facet body with the **Edit Object Display** command and clearing the **Face Analysis** option.

Why should I use it?

You can clean up the color output from the **Detect Primitives** and **Facet Body Curvature** commands so the facet body can better support color coded selection by the **Fit Surface** command.

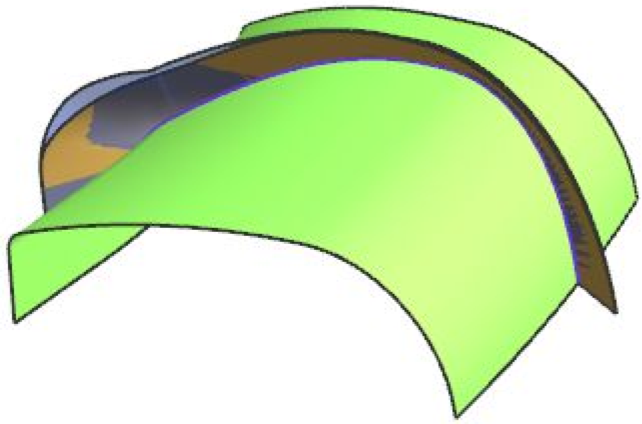
Where do I find it?

Application	Modeling
Command Finder	**Paint Facet Body**
Menu	**Edit→Facet Body→Paint**

Law Extension enhancements

The **Law Extension** command now contains an option to smooth the surface extension.

Use the **Angular Correction** to create a smoother surface.



Where do I find it?

Application	Modeling and Shape Studio
Command Finder	**Law Extension** 
Location in dialog box	**Angle Law** group→**Angular Correction**

Fit Surface enhancements

What is it?

The **Fit Surface** command has been enhanced to improve fitting surfaces to facet bodies and curve features or groups as part of a reverse engineering workflow.

You can now:

- Use color coded selection to select parts of facet bodies for the target.
- Select multiple targets to create multiple Fit Surface features.

Why should I use it?

These enhancements will be useful when:

- You need to fit a face to only a part of a facet body (you no longer need to cut this piece out manually).
- You have multiple shapes of the same type in your target that you want to reverse engineer.

Where do I find it?

Application	Modeling
Command Finder	**Fit Surface**

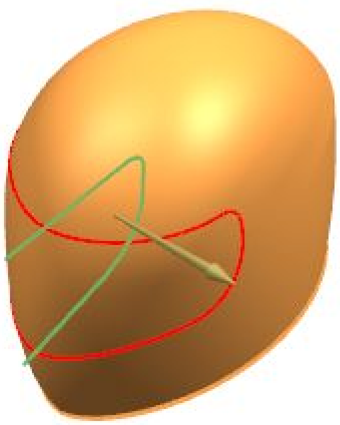
Trim Sheet enhancements

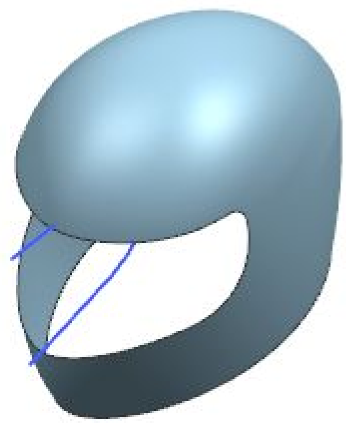
What is it?

The name of the **Trimmed Sheet** command and feature name have been changed to **Trim Sheet**. Legacy **Trimmed Sheet** features will be renamed to **Trim Sheet**. Custom programs that use the feature name **Trimmed Sheet** will need to be updated to **Trim Sheet**.

Additional user interface changes include:

- The Cue line for **Target** selection has changed to **Select sheet bodies to trim.**
- The **Boundary Objects** group name has been changed to **Boundary**.
- The **Cue** line for **Region** has been changed to **Select regions to keep or discard**.





Where do I find it?

Application	Modeling and Shape Studio
Command Finder	**Trim Sheet** 

Synchronous Modeling Blend option

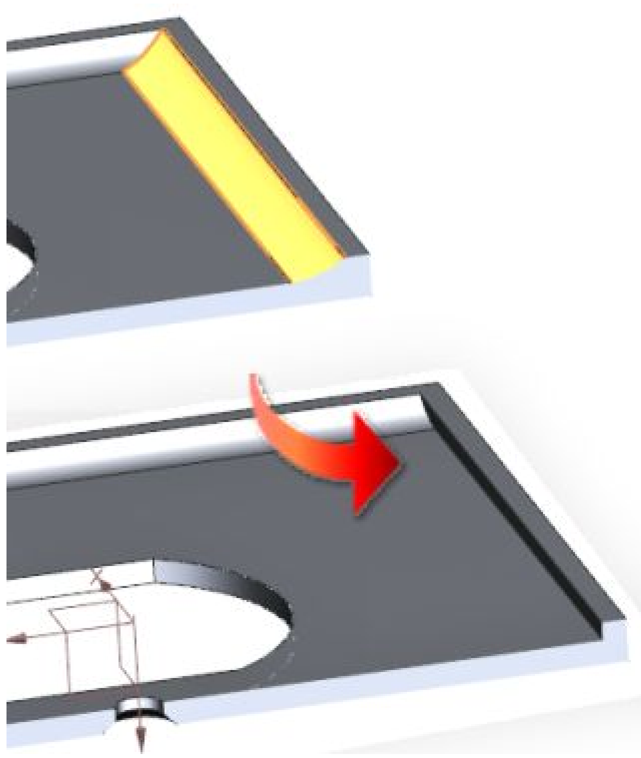
What is it?

The **Delete Face** Synchronous Modeling command includes a new **Blend** type that allows you to delete blends based on the their unique characteristics, such as tangency.

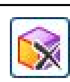
Use this option to delete these types of faces:

- Blends with a constant or variable radius
- Notch blends
- Cliff blends

To delete a notch blend of a cliff blend, use the **Face-Edge Blend Preference** setting. In this example, the blend face was deleted.



Where do I find it?

Application	Modeling
Command Finder	**Delete Face** 

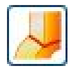
Trim and Extend

What is it?

The **Trim and Extend** command is updated as follows:

- The **By Distance** option has been removed.
- The **Percentage of Measured** option has been removed.
- You may choose from new **Body Output** options: **Extend Original Sheet**, **Extend as New Face** or **Extend as New Sheet**.

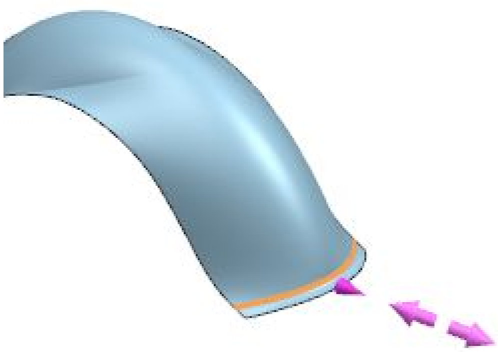
Where do I find it?

Application	Modeling and Shape Studio
Prerequisite	A sheet to trim and additional boundary geometry.
Command Finder	**Trim and Extend** 

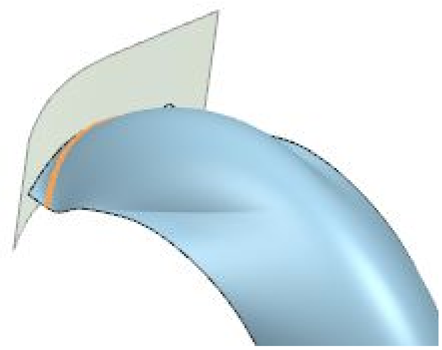
Extend Sheet

What is it?

Use **Extend Sheet** to extend or trim a sheet body. Use **Offset** to trim or extend the sheet at a specified distance from the selected edge. Use **Until Selected** to trim the sheet to other geometry.



Extend using offset



Extend until selected

Additional control includes:

- Surface Extension Shape
- Edge Extension Shape
- Body output type

Where do I find it?

Application	Modeling and Shape Studio
Command Finder	**Extend Sheet** 

Expressions enhancements

The **Expressions** dialog box is enhanced with two new options:

Show Locked Formula Expressions

When selected, expressions with locked formulas display in the expressions list. By default, the check box is unchecked and expressions with locked formulas are hidden.

Replace Expressions

Enables you to replace all instances of an expression formula string within the current work part with another string. This saves you from having to individually edit every expression formula when you have a design change or when you simply want to explore other design scenarios.

Example

You can replace the variable “x” with the variable “y” or with the statement “y+z/2”.

The DesignLogic list is enhanced with two new options:

Expression Expert Mode

Impacts the behavior of NX when you insert a formula in a parameter entry field using the DesignLogic list button .

Expression Expert Mode (checked)	Expression Expert Mode (unchecked)
Length (XC) distance mm Width (YC) 27 mm Height (ZC) distance/2 mm	Length (XC) 100.0000 mm = Width (YC) 27 mm Height (ZC) 50.0000 mm =
The expression formula displays in the field.	The expression value displays in the field.
The formula can be directly edited.	The expression value is locked and cannot be directly edited in the field.

Caution

This optional mode is not intended for general use as it makes it significantly easier for users to inadvertently edit expressions.

You can enable or disable expert mode by setting a customer default called **Expression Expert Mode**.

Extended Text Entry

The **Extended Text Entry** option is added to the DesignLogic list. This option opens the **Extended Text Entry** dialog box, which provides a text entry area where you can enter and edit expression formula text.

Where do I find it?

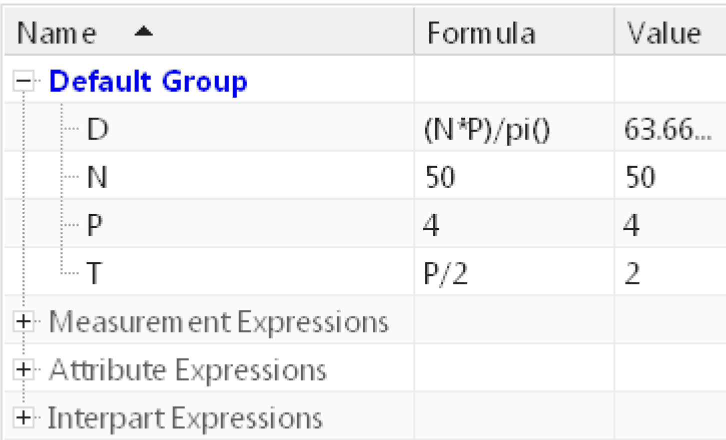
Application	Modeling and Assemblies
Command Finder	**Expressions**

Expression Expert Mode

Command Finder	**Customer Defaults**
Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Simulation→User Interface→Navigator tab→Expression Expert Mode**

Expression groups

You can use expression groups to see an organized hierarchical tree view of expressions in the **Expressions** dialog box, as opposed to a flat list.



Name	Formula	Value
Default Group		
D	(N*P)/pi()	63.66...
N	50	50
P	4	4
T	P/2	2
Measurement Expressions		
Attribute Expressions		
Interpart Expressions		

Name	Formula	Value
Default Group		
D	(N*P)/pi()	63.6...
N	50	50
P	4	4
T	p/2	2
Measurement Expressions		
Attribute Expressions		
Interpart Expressions		

- Only one group is active at a time. Newly created expressions are added to the active group.
- When groups are first enabled, the **Default Group** is active and any existing or newly created expressions reside there.
- Expression groups can contain expressions and nested expression groups.
- Group names must be unique to the part.
- Expressions can only belong to one group at a time. Drag expressions from one group to another.
- Expressions displayed outside of the **Expressions** dialog box do not contain any group information.
- When merging parts, if the source part contains expression groups other than the **Default Group**, those expressions, when transferred from the source part, retain their original group assignments in a new top level group that uses the source part file name as the folder name. Source groups are nested inside.

Why should I use it?

Well-organized expressions are faster to navigate in complex models. Expression groups can enforce design modularity and provide meaningful and descriptive packaging of expressions within models intended for reuse.

Where do I find it?

Application	Modeling and Assemblies
Command Finder	**Expressions**
Location in dialog box	Create or delete a group: With **Show Groups** selected, right-click any group→**New Group** or **Delete Groups** Make active group: With **Show Groups** selected, right-click any group→**Make Group Active**

Interpart expression format

For interpart expressions, a level of indirection has now been created between the expression object and the part name. NX creates a system generated expression for each interpart expression. The system generated expression can then be referenced on the right-hand-side of other expressions.

Note

Existing interpart expressions will be converted to the new format when the part is fully loaded into the NX session.

The **Show Locked Formula Expressions** check box must be selected in order to see the system generated expression.

Where do I find it?

Application	Modeling, Assemblies
Command Finder	**Expressions**
Prerequisite	The **Show Locked Formula Expressions** check box must be selected in order to see the system generated expression.

SNAP

What is it?

Simple NX Application Programming (SNAP) is improved to provide additional capabilities when writing SNAP programs. The major enhancements are:

- Topology functions – These new functions let you get detailed information about the topology of a body (the way in which faces and edges are connected together). The topology functions help you write programs that analyze the shape and structure of solid bodies. For example, you can write programs to find holes, or fillets, or other types of features.

  Also, there are new Shell, Loop, Fin, and Vertex SNAP objects, which extend the existing Body, Face, and Edge topology objects. There is a suite of approximately 20 new enquiry functions that let you find out how these various objects are connected together. For example, you can find all the faces in a shell, all the fins meeting at a vertex, and so on.

- Temporary display functions – These functions let you draw temporary objects in the NX graphics window. You can draw markers, lines, circles, polylines, and text.

  The temporary display functions let you provide graphical prompting or feedback to the users of your programs, which often makes them easier to understand. The user can easily erase the temporary objects after they have served their purpose.

- New documentation – The *SNAP and NX Open for GRIP Users* guide is added that describes SNAP (and to a lesser extent, NX Open) from the point of view of a GRIP user. If you already know GRIP, this guide enables you to easily find corresponding functions in SNAP and NX Open.

In addition, there are other enhancements including new functions to:

- Calculate points on a B-surface very quickly
- Create curves that replicate the shapes of edges
- Analyze the convexity across an edge
- Check whether a point lies inside a body or a face
- Convert between degrees and radians
- Access the NX Work View and other standard NX views

Why should I use it?

You can write programs that can provide additional functionality and features.

Where do I find it?

The *SNAP and NX Open for GRIP Users* guide is in the *Programming Tools* section of the NX Help.

Detailed descriptions of the new functions with examples are in the SNAP Reference Guide. The new functions can be found in the following namespaces and classes:

- Topology objects and functions: new Snap.Topology namespace; additions to Snap.NX.Body, Snap.NX.Face, and Snap.NX.Edge

- Temporary display functions: Snap.Draw class
- Fast calculation of points on a B-surface: Snap.Geom.Surface.Bsurface.Position
- Create curves from edges: Snap.NX.Edge.ToCurve
- Analyze edge convexity: Snap.NX.Edge.Convexity
- Point containment: Snap.NX.Body.ContainsPoint and Snap.NX.Face.ContainsPoint
- Convert between degrees and radians: Snap.Math class
- Work view: Snap.Globals.WorkView
- NX standard views: Snap.NX.View.Standard class

NX Realize Shape

Import Subdivision Geometry

What is it?

Use the **Import Subdivision Geometry** command to import subdivision geometry (not large tessellated data files) which has been exported by other applications using the Wavefront *.obj format.

Why should I use it?

To enable editing subdivision geometry from other applications.

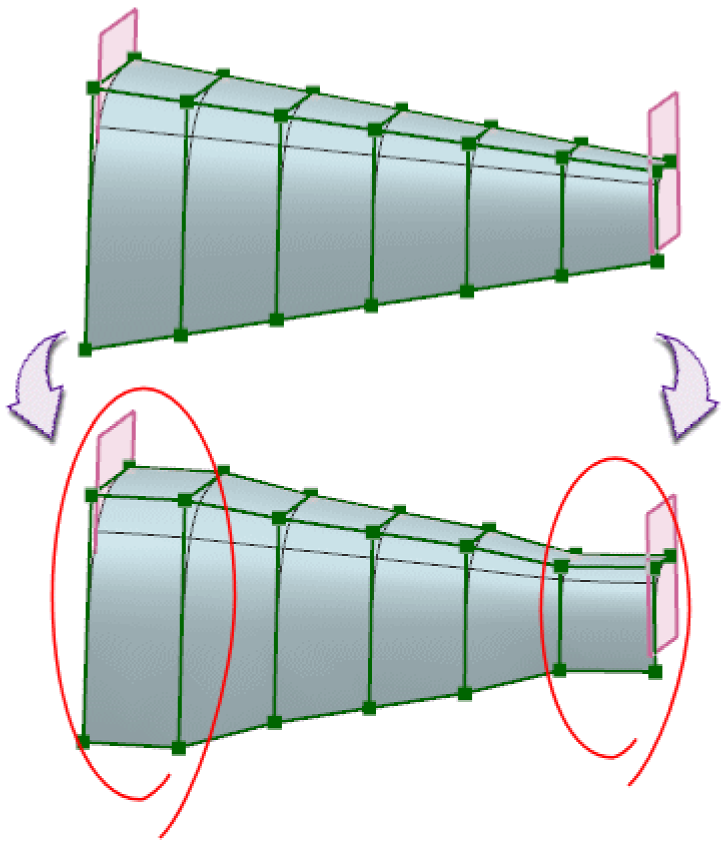
Where do I find it?

Application	Modeling
Command Finder	File Subdivision
Menu	File→Import→Subdivision Geometry

Project Cage enhancements

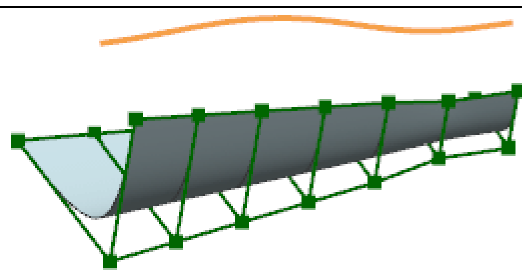
What is it?

The **Project Cage** command is enhanced with the new type **Align Normal**. It aligns selected cage edges with the normals of a selected or defined plane.

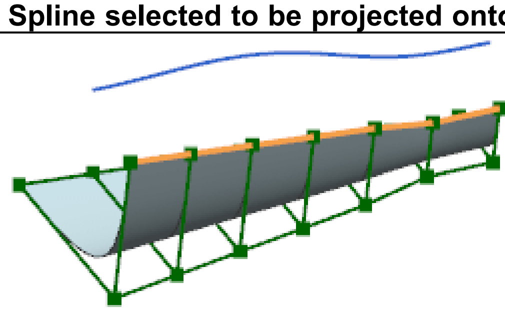


For the **Target Type**, new objects are recognized to be projected onto:

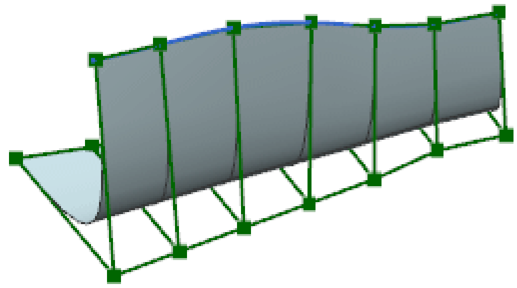
Arcs and **Circles** are now recognized as objects.
Vertices of selected cage edges are projected onto the selected arc or spline



Spline selected to be projected onto



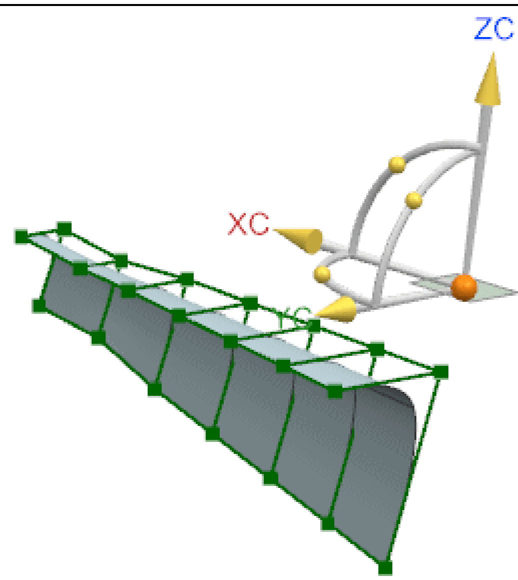
Edges selected to project



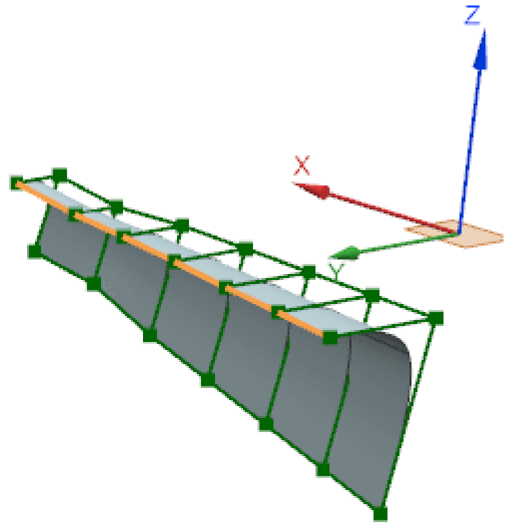
Vertices of selected edges now on spline

Dynamic Coordinate System

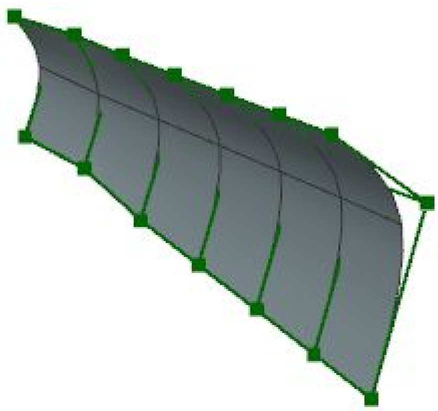
Vertices of selected cage edges are projected onto the XY plane of the dynamic coordinate system. You can manipulate the dynamic coordinate system so its XY plane is the desired plane to project to, prior to clicking Apply or OK



XY plane of dynamic coordinate system oriented as desired



Edges selected to be projected



Vertices of selected edges now on XY plane of dynamic coordinate system

Why should I use it?

To make cage edges linear or parallel and/or to prepare for a high quality symmetric model.

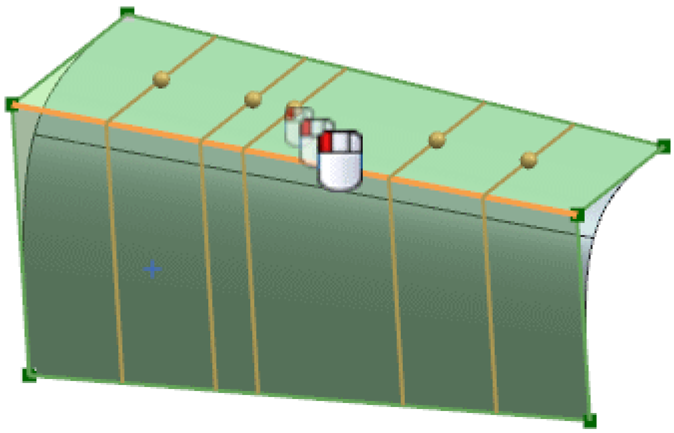
Where do I find it?

Application	Modeling and Shape Studio
Task Environment	NX Realize Shape
Command Finder	**Project Cage** 
Location in dialog box	**Type** group

Split Face enhancements

What is it?

- The **Uniform** type now allows you to drag the split lines during creation. Handles are provided for this at creation time.



- The **Along Polyline** type now allows you to select **Point on Face**.

Where do I find it?

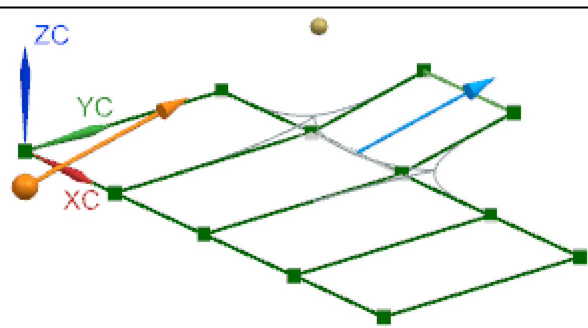
Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Split Face**
Location in dialog box	**Type** group, **Uniform**

Extrude Cage enhancements

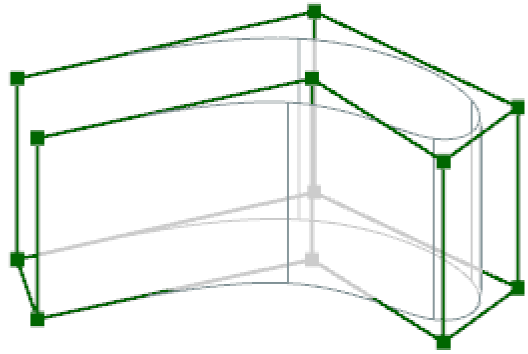
What is it?

The **Extrude Cage** command is enhanced to let you:

- Specify an extrude direction for selected faces, edges, and NX Realize Shape.

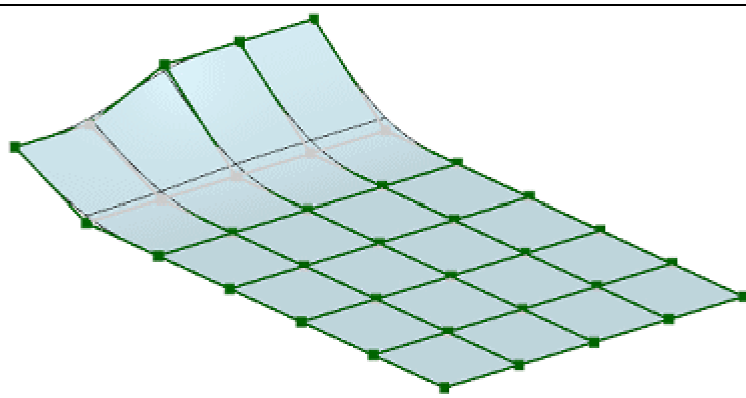


Single edge extruded at an angle

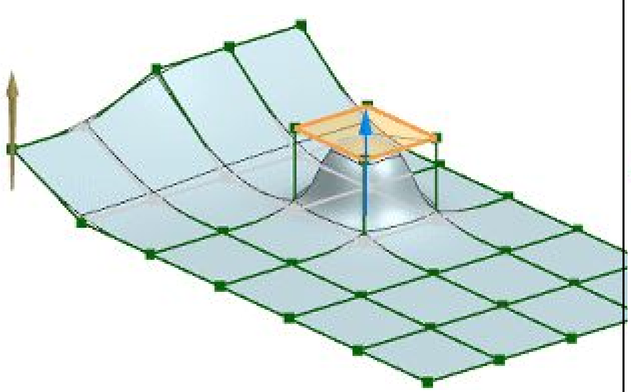


Polyline extruded

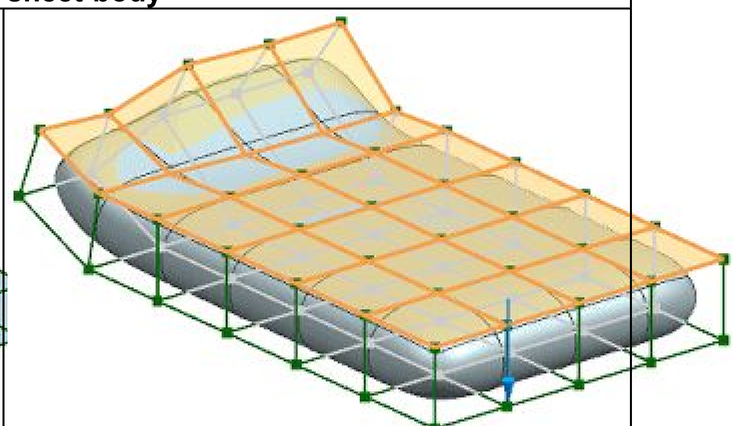
- Use internal edges as support.
- Add additional sections in the extrusion direction (polylines and internal edges only).
- Optional conditional thickening of open sheet bodies. It must be an open sheet body, and all the faces must be selected.



Open edge sheet body



One face extruded—no thickening



All faces selected—thickened

Why should I use it?

Use it when you have polylines to be extruded, or you need to extrude a polyline, edge or face in a specific direction.

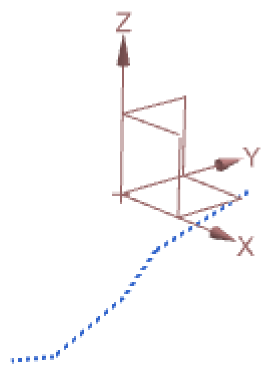
Where do I find it?

Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Extrude Cage** 

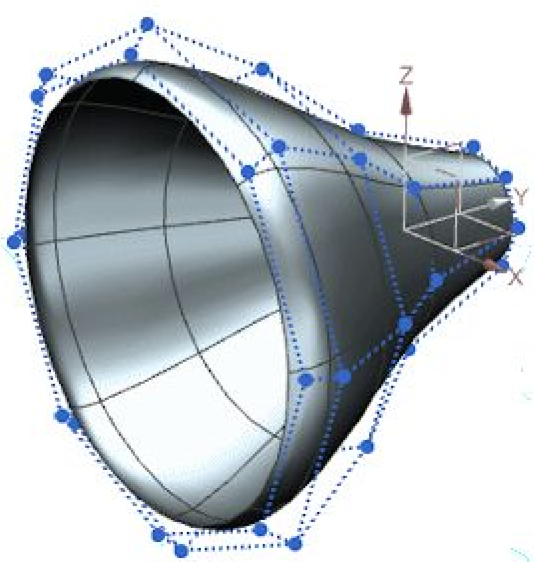
Revolve Cage

What is it?

Use the **Revolve Cage** command to revolve a selected set of polylines or cage edges about an axis to generate a control cage.



Polyline to be revolved



Polyline revolved about the Y axis 360 degrees, with eight segments specified

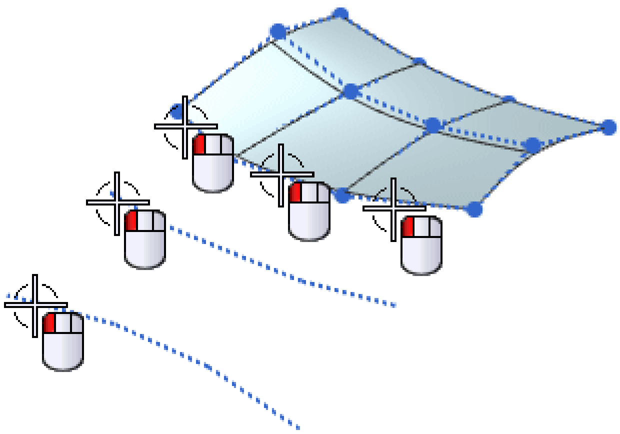
Where do I find it?

Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Revolve Cage**

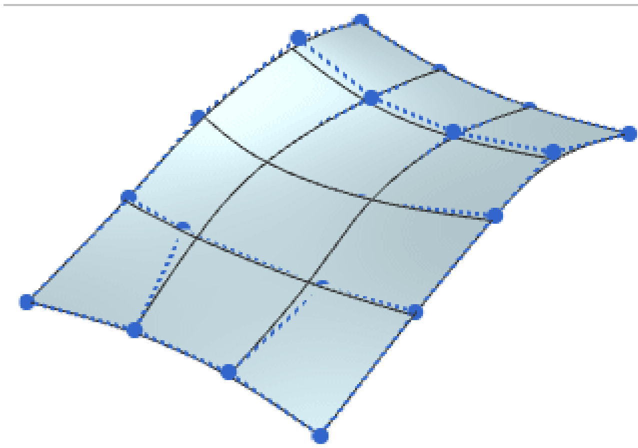
Loft Cage

What is it?

Use the **Loft Cage** command to create a loft of control cages using sets of face edges and polylines.



Two three-segment polylines and 3 cage edges selected to be lofted



Loft cage created, with one segment specified

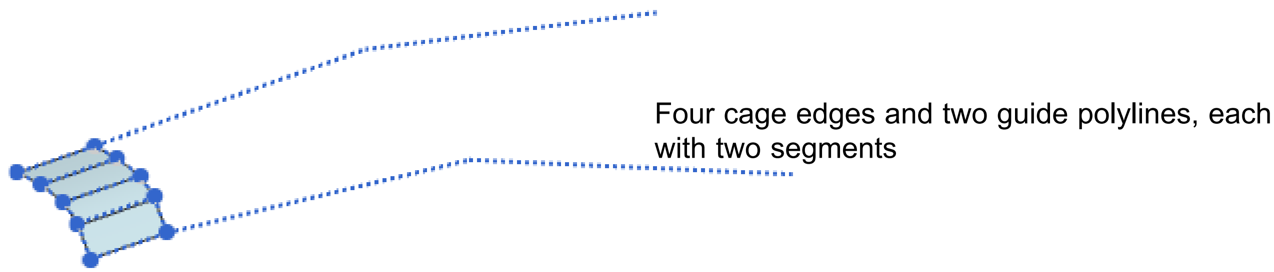
Where do I find it?

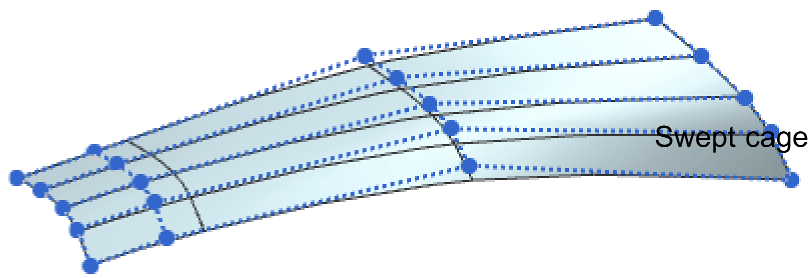
Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Loft Cage**

Sweep Cage

What is it?

Use the **Sweep Cage** command to create a control cage and a subdivision body when you have section polylines or control cage edges and no more than two guide polylines or edges.





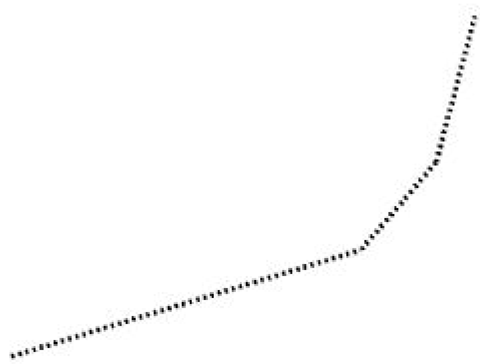
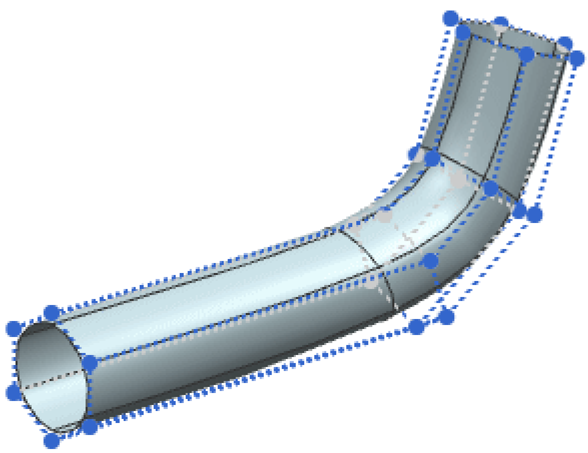
Where do I find it?

Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Sweep Cage**

Tube cage

What is it?

Use the **Tube Cage** command to create a tube using a polyline as a centerline and by specifying the number of cage segments.

	Three-segment polyline
	Tube with 6 segments specified

Where do I find it?

Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Tube Cage** 

Transform Cage enhancements

What is it?

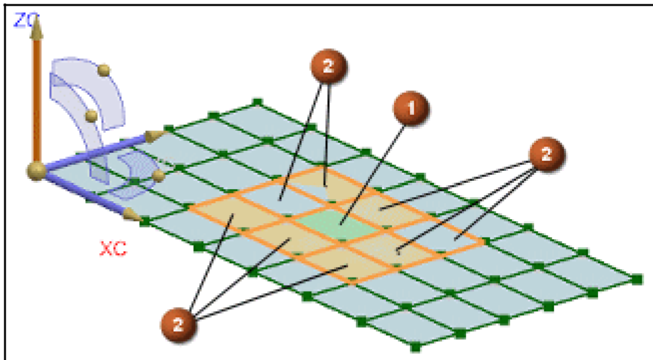
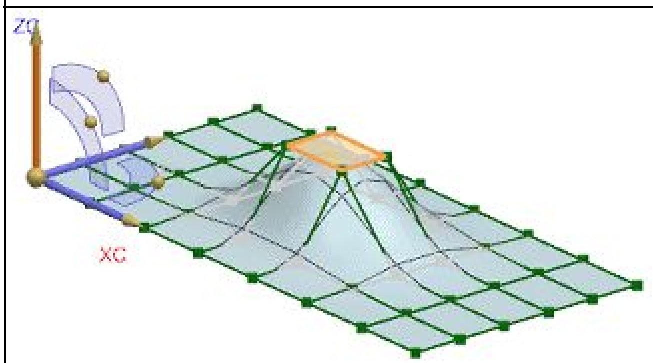
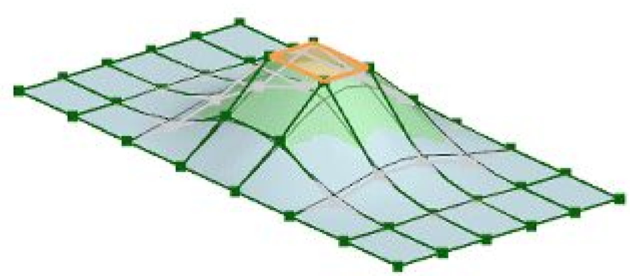
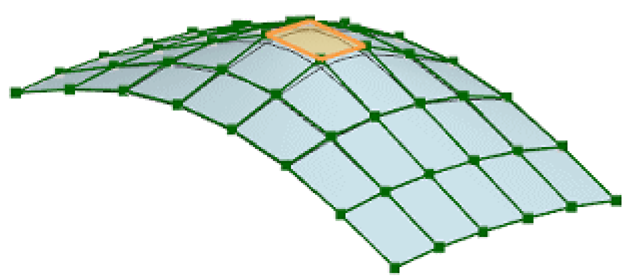
The **Transform Cage** command has been enhanced with a new falloff option. This option gives you a smooth transition between the transformed objects and the remaining objects.

The user can choose one of three options:

- **None**–(the default)
- **Selected**–The user-selected objects which should participate in the falloff.
- **All**–All remaining objects will participate in the falloff.

The effective falloff transformation depends on the distance to the primarily transformed objects.

The **Falloff Curve** can be set to emphasize closer objects, be linear, or can be set to emphasize distant objects.

	Falloff selections: 1. Selected Object 2. Selected Falloff Objects
	Falloff selected: **None**
	Falloff selected: **Selected**
	Falloff selected: **All**

Why should I use it?

When you want a more gradual effect on a transformation.

Where do I find it?

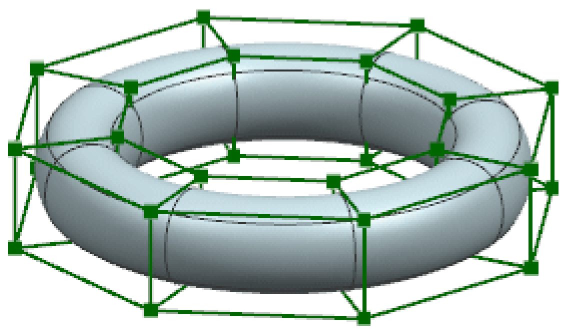
Application	Modeling and Shape Studio

Task environment	NX Realize Shape
Command Finder	**Transform Cage**
Location in dialog box	Below **Method**

Primitive Shape enhancement

What is it?

The **Primitive Shape** command is enhanced to include the **Torus** shape type.



Why should I use it?

Use **Torus** if you want to create surface subdivision geometry that is toroidal in shape.

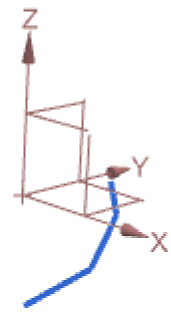
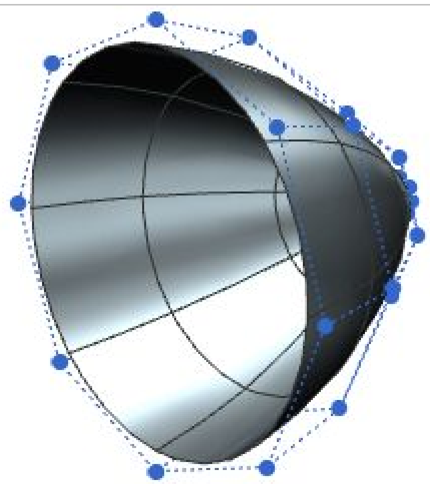
Where do I find it?

Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Primitive Shape**
Location in dialog box	**Type** group

Cage Polyline

What is it?

Use the **Cage Polyline** command to create or edit a line or series of lines used to create control cages.

	A three segment polyline selected to be revolved
	Polyline revolved 360 degrees, in eight segments

Use **Cage Polyline** to:

- Create a polyline by specifying its points. Each point specified defines a segment of the polyline.
- Select any point you want, on cage geometry or non-cage geometry.
- During creation, drag the location of any polyline segment end point.

Why should I use it?

Use **Cage Polyline** as a preparation for extruding, revolving, lofting, or sweeping them into cage faces,.

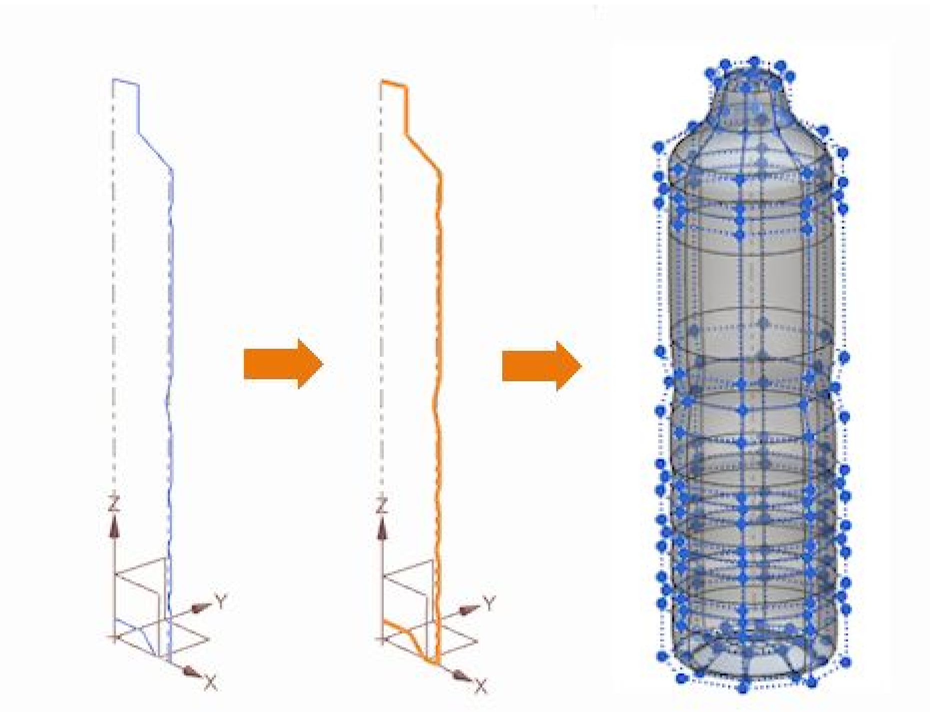
Where do I find it?

Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Cage Polyline**

Extract Cage Polyline

The **Extract Cage Polyline** command lets you extract cage polylines from an existing curve or face edge from outside a subdivision feature.

- Each linear section is one polyline segment.
- A non-linear section is broken into as many linear segments as you specify.

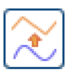


In the example above, a polyline is extracted from sketch geometry, then revolved to create a bottle.

Why should I use it?

Use the **Extract Polyline** command to create polylines from non-cage geometry. The extracted polylines are then used to generate cage geometry using **Extrude Cage**, **Revolve Cage**, **Loft Cage**, **Sweep Cage**, or **Tube Cage**.

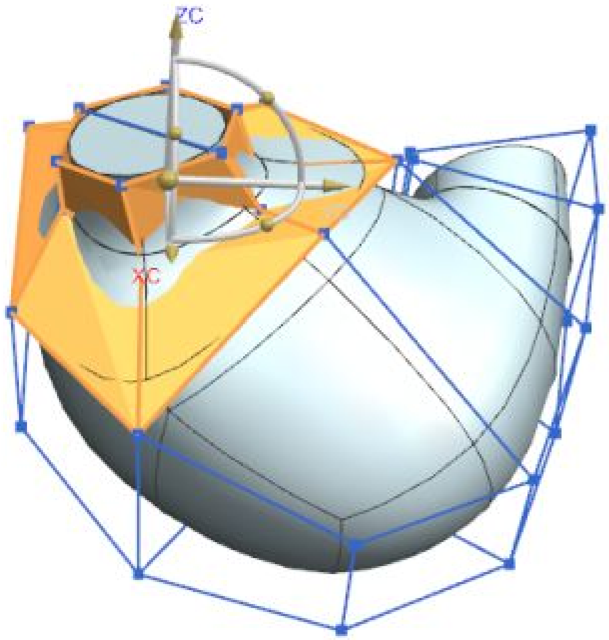
Where do I find it?

Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Extract Polyline** 

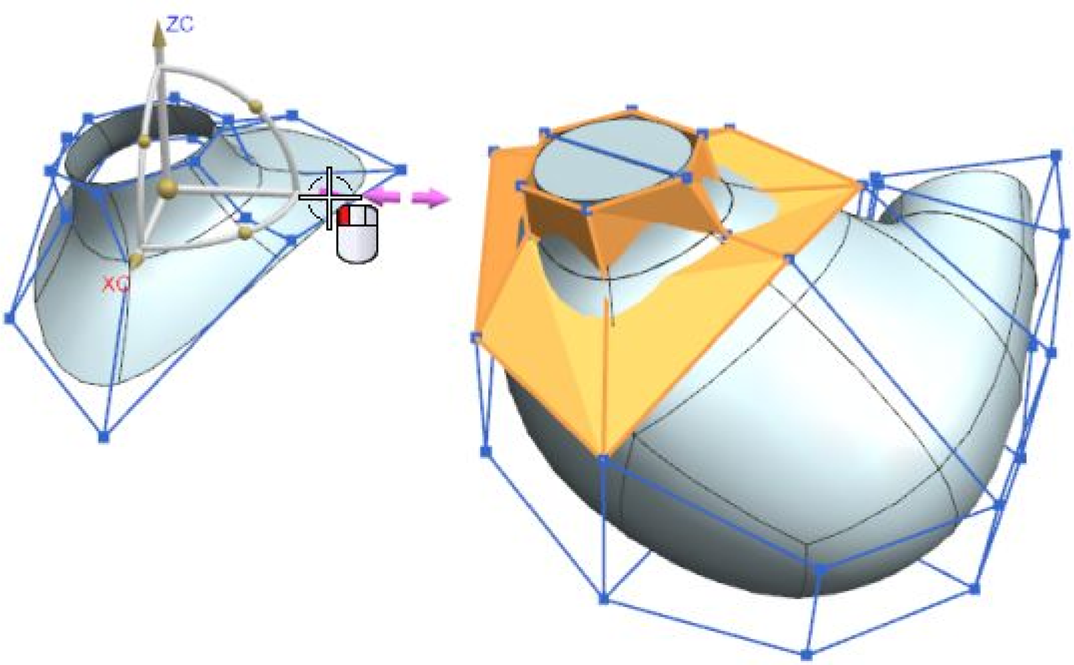
Copy Cage

Use the **Copy Cage** command to copy and move selected cage faces and polylines.

- Selected cages that you want to move and copy highlight, and the *cage manipulator* is positioned to the averaged center of the selected elements.



- Use the *cage manipulator* to specify the translation and rotation movement you want.



- Each copy of the selected cages creates new topology, unassociated to the parent cage, within the subdivision feature.

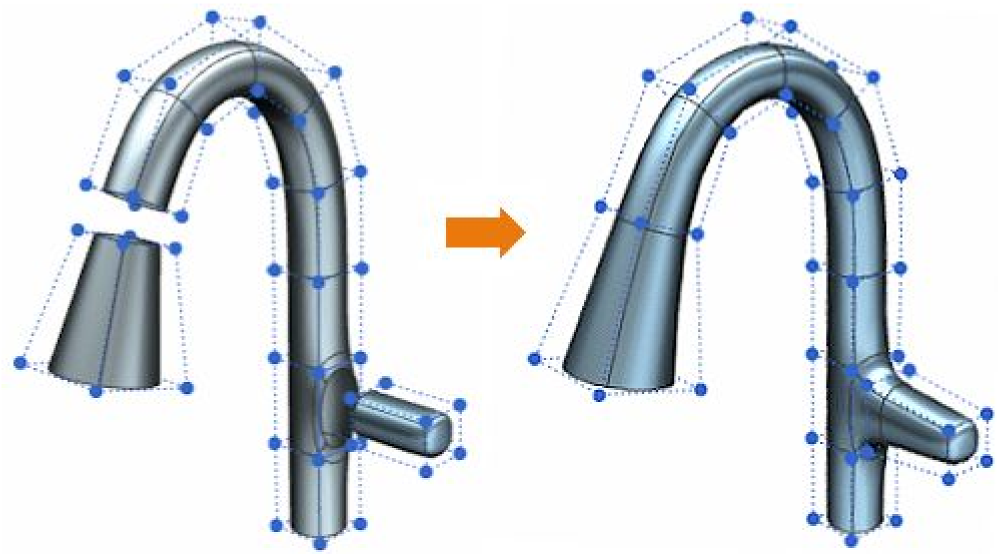
Where do I find it?

Application	Modeling and Shape Studio
Task environment	NX Realize Shape
Command Finder	**Copy Cage**

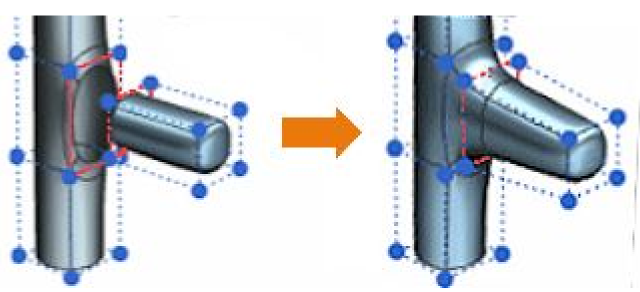
Sew Cage

What is it?

Use the **Sew Cage** command to connect open control cage edges of a subdivision feature.



- You select two sets of open edges to be connected. You can select the edges directly or by selecting cage faces near an open edge.
  - o Each set needs the same number of edges.
  - o You can select edges in any order.
- When there are gaps, the selected cage edge pairs are averaged. The continuity setting of the connecting surface is inherited from the target edge.



Where do I find it?

Application	Modeling
Task environment	NX Realize Shape
Command Finder	**Sew Cage** 

NX Realize Shape preferences enhancements

What is it?

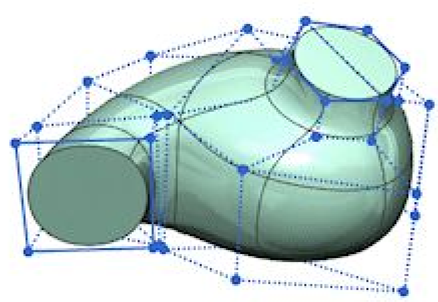
The preferences for **NX Realize Shape** now include options to control cage display, enhance cage selection and display weights.

 **Cage and Body**

There are three different display modes when editing subdivision feature within the task environment.

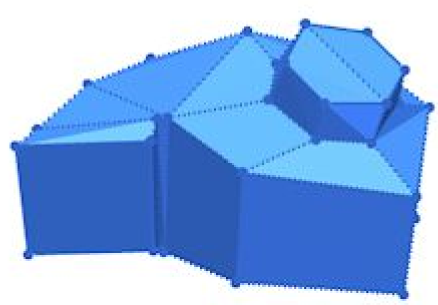
Cage and Body

This is the default display and most commonly used display mode.



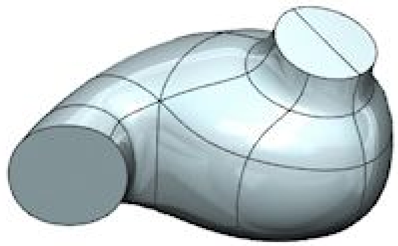
Cage Only

Use the **Cage Only** display mode when you want to create the main structures of a shape without waiting for body display updates. The cage faces are shaded in this mode.



Body Only

Use the **Body Only** display mode when you want to do surface analysis of the resulting body without exiting the task environment. Due to the hidden control cage, editing is not possible in this mode.



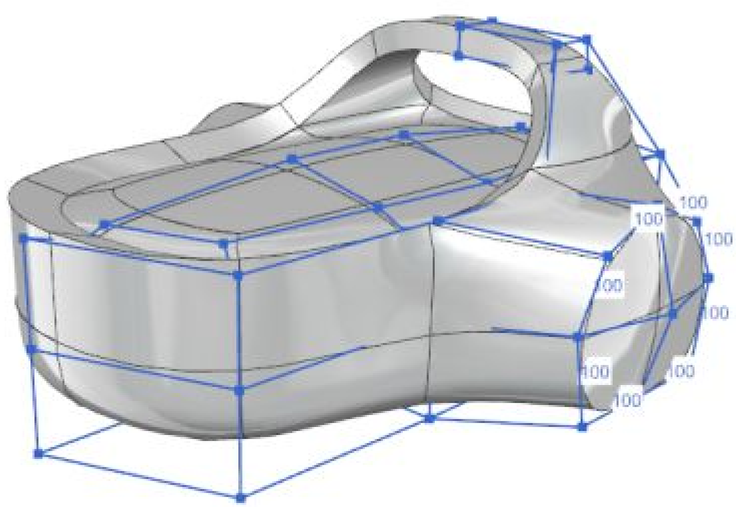
 **Allow Backside Selection**

Use this as a selection filter when selecting cage elements.

- When active, you can select cage elements that lie behind other subdivision geometry. This option is on by default.
- When inactive, you cannot select any cage elements on the back side of control cage, even if you can see the element through the transparent cage.

 **Show Weight**

Use the **Show Weight** display mode when you want to display the current weight factors set on cage elements.



NX Realize Shape Preferences

Use the **NX Realize Shape Preferences** dialog box to set control cage display preferences. You can set:

- The cage display mode preferences.

  Cage and Body **Cage Only** **Body Only**

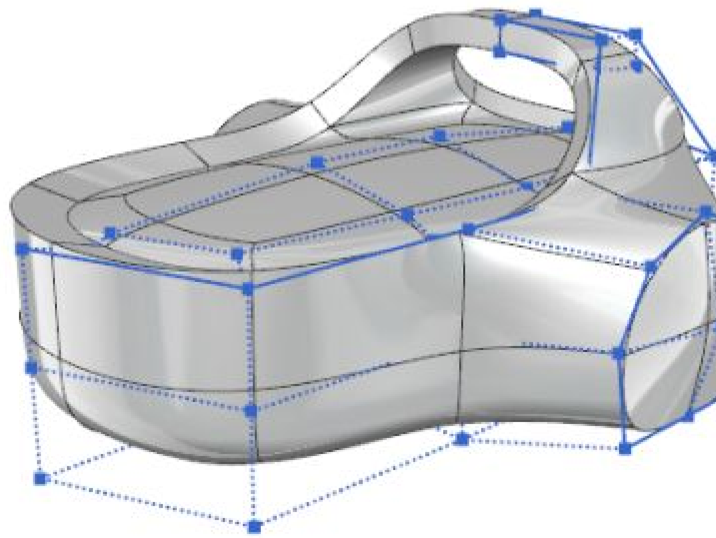
- The cage object display preferences.

  Show Weight

  Line fonts for display of sharp and smooth edges.

  Line Font (Smooth Edge) = Dotted

  Line Font (Sharp Edge) = Solid



- The control cage selection preferences.

  Allow Backside Selection

Where do I find it?

Application	Modeling
Task environment	NX Realize Shape
Command Finder	**Cage and Body**, **Allow Backside Selection**, **Show Weight**, and **NX Realize Shape Preferences**

Assemblies

Visual Reporting

Create an HTML report for Visual Reporting

What is it?

Use the new **Create HTML Report** option to send the results from an activated visual report to an HTML web page. You can set the **Create HTML Report** option so that when you run a visual report you create an HTML web page containing the results.

In the report, you can browse the results tree and its corresponding **Info View** information.

Where do I find it?

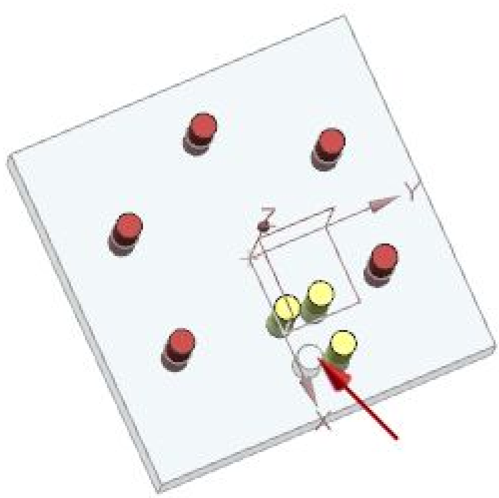
Prerequisite	**Legend Style** must be set to **Tree**, and you must run a visual report.
Resource Bar	**HD3D Tools→Visual Reporting**
Location in dialog box	**Results** group→right-click the tree background→**Create HTML Report**

Creating visual reports for component patterns

What is it?

Use the new **Component Pattern Type** option to define a visual report that reports on the component patterns in your assembly.

The following example shows the results of running a visual report using the **Component Pattern Type** option. The arrow points to the master component. The circular pattern components are red and the linear pattern components are yellow. The reference pattern color, blue, does not appear because its components are the same ones that are used in the circular array



Where do I find it?

Prerequisite	In the **Visual Report Definition** dialog box, in the **Report Property** group, you must set **Source** to **Component Property**. **Note** You can open the **Visual Report Definition** dialog box by clicking **Define New Report** in the HD3D Visual Reporting tool.
Location in dialog box	**Report Property** group→**Property**→**Component Pattern Type**

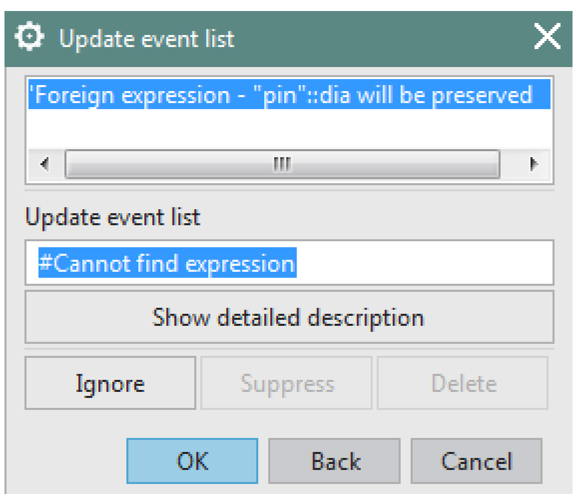
Preserving broken interpart expressions

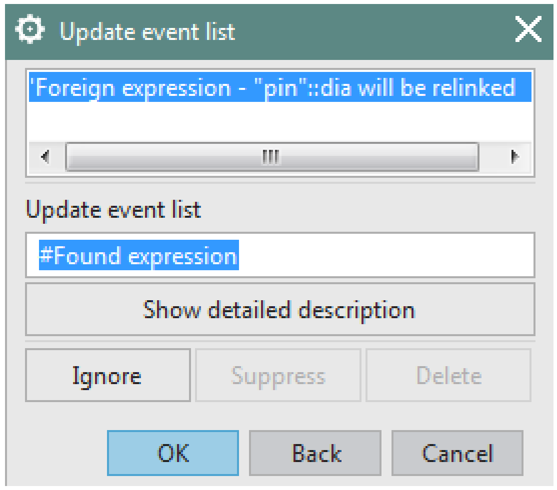
What is it?

Interpart expressions are now preserved when they break. An interpart expression will break when the source part file or the expression in the source part file cannot be found. In previous versions of NX, the interpart formula would be replaced by the last known value. If the source part file or the expression in the source part file was found, then the interpart expression would have to be constructed again.

In NX 10.0.1, if an interpart expression is broken, NX preserves the interpart expression formula and marks the expression as broken. If the source part file is found or if an expression with the correct name is created in the source part file, then the interpart expression is relinked automatically. NX retains the last known value for the expression as long as it remains broken.

The user is informed when the interpart expression's condition changes to either preserved or relinked.





Why should I use it?

Preserving the formula describing the interpart expression and marking the expression as broken retains more of the designer's original design intent and saves time and effort in finding and repairing broken interpart expressions. Automatic relinking similarly removes the need to recreate broken interpart expressions when a valid parent is reintroduced.

Where do I find it?

Application	Modeling
Command Finder	**Expressions**

Mirror Assemblies Wizard

What is it?

If you are in a Teamcenter environment, you now use the business logic set up for your site when you create new mirror parts using the **Mirror Assembly Wizard** (MAW). The MAW also supports the new Teamcenter multi field key attributes.

The new items created in Teamcenter for any newly created mirror parts will be of the same type as the source parts being mirrored. When you mirror a part or assembly, you create a new part or assembly, not a new revision.

Note

You can no longer change the item type for mirrored parts with the **Mirror Assembly Wizard**.

Why should I use it?

You can rename all the new objects and their attributes, including Teamcenter multi field attributes, at one time in a single mirroring process. You can see and validate all the attributes as you mirror them.

Where do I find it?

Prerequisite	You must be in a Teamcenter environment.

Command Finder	**Mirror Assembly**

Extraction Path customizing

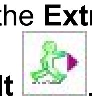
What is it?

When creating an extraction path in assembly sequencing, use the new **Guide Path Manually** option to influence the direction of the path that is created. This lets you drag a copy of the components to a location in the extraction path. NX then attempts to calculate and create a valid path in that direction.

Why should I use it?

The **Guide Path Manually** option is useful when the extraction path may be complicated by the presence and location of other components. By guiding the components in a recommended direction, you provide NX with information that helps define a path that would otherwise be time-consuming or difficult to find.

Where do I find it?

Application	Assemblies
Command Finder	**Extraction Path**
Location in dialog box	Located in the version of the **Extraction Path** dialog box that appears after you click **Generate Result** .

Replacing a part families component

What is it?

The Part Families functionality has been enhanced to ensure that drawing data (such as dimensions, occurrence notes, callouts) and occurrence properties (such as color and visibility settings) are preserved in certain situations when creating assembly part family members. Prior to this change, drawing data would be retained and occurrence properties deleted when the assembly part family member creation required some component of the template assembly to be substituted with another component. This data loss would occur even when the substituted parts were versions of each other.

Why should I use it?

This enhancement is relevant to users that require better associativity support when creating assembly part family members.

Where do I find it?

Application	Assemblies
Command Finder	**Part Families**
Menu	**Menu→Tools→Part Families**

Default location for component drag handles

What is it?

You can now specify a default location for the drag handles of components that do not have a user-defined drag handle location. To do this, set the **Drag Handle Location** customer default to either **Center of Bounding Box** or to **Origin of Component**.

For parts modeled at the global origin, the component origin is a preferred default location for a drag handle. For parts modeled in their location in the assembly, the origin of a component might be quite a distance from the part geometry. In these cases, a CSYS or datum CSYS on the part is a preferred location.

You can now use the following drag handle shortcut menu options to quickly snap the drag handle to the desired location:

- **Snap Handle to CSYS**
- **Snap Handle to Component Center**
- **Snap Handle to Component Origin**

If you select multiple components, the default drag handle location is always at the center of the combined bounding box of all selected components.

Note

A user-defined drag handle location takes precedence over the location set by the customer default.

Why should I use it?

When you design a manufacturing line within a plant, you can quickly and precisely place components based on their origin or a CSYS.

Where do I find it?

Command Finder	**Customer Defaults**
Location in dialog box	**Assemblies→Positioning→Interface** tab

Command Finder	**Assemblies Preferences**

Moving a component along a vector and projected distance

What is it?

You can now move a component along a vector and by a projected distance between two objects or points that are projected on the motion vector. To do this, set the **Motion** type option to **Projected Distance**, and set the following options:

- **Specify Vector**: Lets you specify a vector for the projected axis.
- **Select Start Point or Object**: Lets you select the point from which to measure the distance.

- **Select End Point or Object**: Lets you select the end point for measuring the distance.

Where do I find it?

Command Finder	**Move Component**
Location in dialog box	**Transform** group→**Motion** **Transform** group→**Specify Vector** **Transform** group→**Select Start Point or Object** **Transform** group→**Select End Point or Object**

Add a component to the XC-YC plane by drag and drop

What is it?

You can use the **Position on XC-YC Plane** customer default to enable quickly placing of new components by drag and drop to the XC-YC plane.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

Why should I use it?

To quickly place components at Z=0 to simulate objects being placed on a floor.

Support for monolithic JT files

What is it?

A monolithic JT file is a JT file that contains the entire product structure. When you add a monolithic JT file as a component to an NX assembly, NX now creates JT faceted bodies. In previous releases, NX created legacy NX faceted bodies.

WAVE-linking product interface enhancement

What is it?

When you use the graphics window to select an object that has a product interface, the product interface is now selected for the following WAVE commands:

- **WAVE Geometry Linker**
- **Copy Geometry to Component**
- **Copy Geometry to Part**
- **Copy Geometry to New Part**

When you use one of the copy commands, the object you select must correspond exactly with a product interface in order to link to the product interface. The WAVE mode object types that correspond with product interface object types are:

- Body
- Datum
- Point
- Sketch
- Composite curve

Note

Edges and faces do not link to a product interface because those WAVE mode object types do not correspond to product interface object types.

Why should I use it?

The behavior of these commands is now consistent with the behavior of other WAVE commands. Previously, when you used the graphics window to select geometry that has a product interface, some WAVE commands selected the product interface while others selected the object geometry. Most users prefer to link to the product interface instead of the geometry because, if the product interface is remapped to new geometry, the associated WAVE link also updates to point to the new geometry.

Where do I find it?

WAVE Geometry Linker

Command Finder	**WAVE Geometry Linker**

Copying geometry

Prerequisite	Make sure that **WAVE Mode** is active in the **Assembly Navigator**.
Assembly Navigator	Right-click on a component→**WAVE**→**Copy Geometry to Component**, **Copy Geometry to Part**, or **Copy Geometry to New Part**

Updating assembly structures in the Assembly Navigator

What is it?

The **Update Structure on Expand** customer default check box is now selected by default. This ensures the following occurs when you expand nodes for unloaded subassemblies in the **Assembly Navigator**:

- The expanded assembly structure is up-to-date.
- The bounding boxes are correct.
- The attributes are up-to-date.

Update Structure on Expand has no effect on loaded subassemblies, because their structure, bounding boxes, and attributes are updated as part of the load process.

You can select or clear the **Update Structure on Expand** preference check box to experiment with its effects on performance in your NX session.

Why should I use it?

If the **Update Structure on Expand** check box is not selected, unloaded assemblies could display out-of-date information when you expanded their **Assembly Navigator** nodes.

Where do I find it?

Update Structure on Expand customer default

Ribbon bar	**File** tab→**Utilities**→**Customer Defaults**
Location in dialog box	**Assemblies**→**General**→**Miscellaneous** tab→**Update Structure on Expand**

Update Structure on Expand preference

Menu	**Menu**→**Preferences**→**Assemblies**→**Update Structure on Expand**

Interpart expression format

For interpart expressions, a level of indirection has now been created between the expression object and the part name. NX creates a system generated expression for each interpart expression. The system generated expression can then be referenced on the right-hand-side of other expressions.

Note

Existing interpart expressions will be converted to the new format when the part is fully loaded into the NX session.

The **Show Locked Formula Expressions** check box must be selected in order to see the system generated expression.

Where do I find it?

Application	Modeling, Assemblies
Command Finder	**Expressions**

Prerequisite	The **Show Locked Formula Expressions** check box must be selected in order to see the system generated expression.

Drafting

Designing using Layout

Layout process in NX

Layout describes a conceptual 2D design environment where an engineer or designer can explore concepts in 2D by creating layouts of a part or assembly. The task is not to produce a finished drawing, but to provide an alternate way of solving design problems whose key characteristics are represented in 2D. You can use it for rapid prototyping of a design or part.

The layout process lets you do concept design without modeling 3D geometry, but it achieves the requirements that will drive the 3D design.

In NX, the **Layout** tab centralizes main commands in the following groups for exploring concepts in 2D.

- **View** commands
- **Sketch** and **2D Component** commands
- **Dimension** and **Annotation** commands

2D components are rigid reusable objects dedicated for 2D design that you can use to quickly and easily create a layout. In the context of the 2D design, 2D components are analogous to 3D parts.

Using the layout process, you can:

- Create and organize 2D components into a hierarchical structure in the **Part Navigator**.
- Edit an active 2D component including nesting other 2D components. Other 2D components with same definition can be immediately updated or you can delay the update and lock 2D components to not apply the changes.
- Drag and drop from the **Reuse Library** to add existing 2D components to your layout.
- Apply different settings to each 2D component to manage the visibility of 2D components, geometry, dimensions, and annotations or apply them to all 2D components.
- Replace existing 2D components with one other 2D component.
- Break associativity between the 2D component and its definition or smash it into its constituent objects.
- Document your layout for design review or production.
- Move a 2D component to a new view.
- Copy 2D components to 3D modeling space or create an assembly to continue your design in 3D.

- Publish 2D component definitions to Teamcenter or a disk drive for sharing with other engineers.

2D components and sketch integration

2D component instances are sketch objects that you can manipulate and position as any other sketch object. Their geometry is rigid and changes only when edited.

2D component hierarchy is shown in the **Part Navigator** under the sketch node.

Drawing
  Sheet “Sheet 1” (Work-Active)
    Sketch “SKETCH_SHEET 1_000” (Active)
      2DComp
  Out-of-Date

You can apply sketch geometric and dimensional constraints:

- On 2D component geometry.
- Between 2D component instances.
- Between 2D component geometry and sketch geometry.

When you edit a 2D component, NX preserves the constraints as long as, during the edit, you do not delete the 2D component geometry where the constraint is placed.

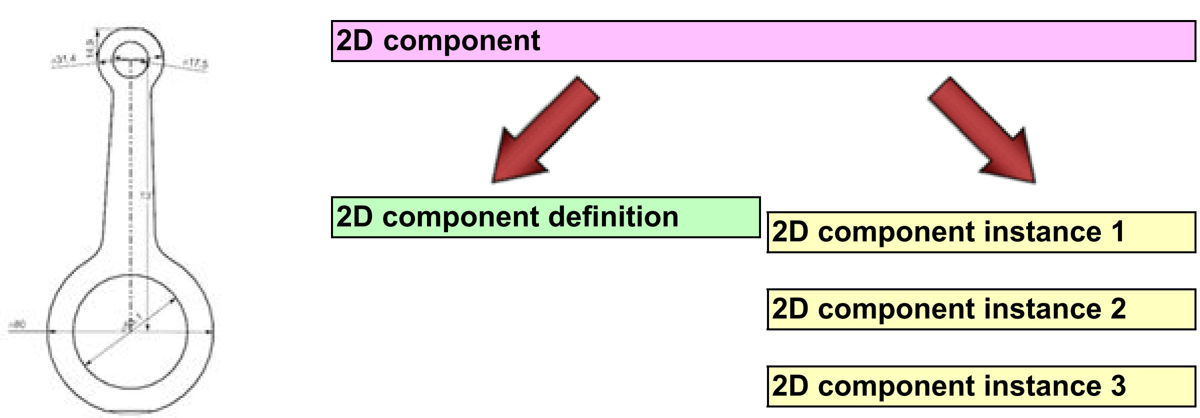
Note

A 2D component can snap to the grid and supports **Move**, **Rotate**, **Offset**, **Cut**, **Copy**, and **Paste** commands.

2D components

A *2D component* is a reusable drafting object dedicated for 2D design and layout. It is analogous to a 3D part in the context of a 2D environment.

Each 2D component has one *2D component definition* and one or more *2D component instances*.



2D component definition	Defines the 2D component recipe that contains the geometry, the name, the anchor point, and a preview image of the 2D component. It is stored in the **Reuse Library**. You can place multiple instances of the 2D component definition in your layout. When the 2D component definition changes, all of its instances are updated.
2D component instance	Represents the 2D component in your layout and displays it in the graphics window. A 2D component instance is a sketch object that belongs to a sheet sketch, view sketch, or another 2D component. It is associatively linked to the 2D component definition. Each 2D component instance has a unique name derived from the 2D component definition name.

Supported 2D component objects

The supported objects for a 2D component are:

- Sketch objects
- Drafting and sketch dimensions
- Drafting annotations including custom symbols
- Centerlines
- Nested 2D components

A 2D component is another sketch object; as such, it cannot contain basic curves, drafting curves, 3D solids, 3D faces, 3D edges, groups, drawing views, tables, title blocks, and images. In addition, a 2D component cannot contain sketch groups and a sketch group cannot contain 2D components.

2D component Reuse Library

All 2D component definitions are stored in the **Reuse Library**. The definitions can be stored in the current part file, in a centralized library created on a disk drive, or in Teamcenter.

Reuse Library

- 2D Component Library
  - Published 2D Components
    - **Simple 2D Components**
    - **Complex 2D Components**
  - Local 2D Components

2D components stored in the current part file are called *local 2D components*. They appear in the **Local 2D Components** folder node under **2D Component Library**.

2D components stored in a centralized library on a disk drive or in Teamcenter are called *published 2D components*. They appear in the **Published 2D Components** folder node under **2D Component Library**.

The name and location of the centralized 2D component library is defined in **Customer Defaults**. You can create subfolders for managing your 2D component definitions.

2D component settings and preferences

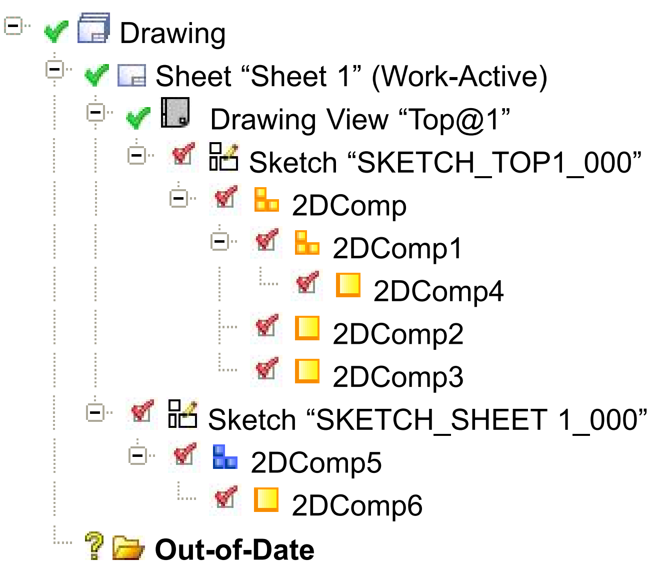
You can modify the appearance of one or more 2D component instances through the **Settings** dialog box. To modify the appearance of all newly created or placed 2D components, use the **Drafting Preferences** dialog box.

You can show or hide:

- Reference geometry
- Annotations
- Dimensions

New 2D Component

Use the **New 2D Component** command to create an empty 2D component instance and store its definition in the **Reuse Library**. You can continue to create additional empty 2D components including nested 2D components to build a virtual structure in the **Part Navigator** before adding geometry and annotation to the 2D components.



When you use the **New 2D Component** command, NX adds the newly created 2D component to the active sketch. The active sketch belongs to one of the following:

- Active sheet
- Active drawing view
- Active 2D component

As you are creating the 2D component, you can rename it or use the uniquely generated name. You can modify the prefix NX uses with the 2D component **Name Prefix** customer default.

You must make the newly created 2D component active before adding geometry and annotation to it.

Where do I find it?

Application	Drafting
Command Finder	**New 2D Component**
Graphics window	Right-click the 2D component→**New 2D Component**
Part Navigator	Right-click the sheet node, the drawing view node, the sketch node, or the 2D component node→**New 2D Component**

Define 2D Component

Use the **Define 2D Component** command to define a 2D component instance by selecting its content in the graphics window and storing its definition in the **Reuse Library**.

You can select any existing sketch geometry including other 2D components, annotations, and dimensions from the active sketch. The active sketch can be located in the sheet, the drawing view, or the active 2D component.

In addition to the selection, in the **Define 2D Component** dialog box, you can specify:

- The anchor point.
- The location in the **Reuse Library** where you want to store the new definition.
- The descriptive name of the 2D component definition.
- The preview image.

NX creates a 2D component definition in the **Reuse Library** that contains all copied sketch geometry including constraints and nested 2D components. If the selection contained a sketch group, the content of the sketch group is moved to the 2D component. The group stays empty.

In the graphics window, NX replaces the selected objects by the 2D component instance of the newly created 2D component. All constraints and dimensions between the 2D component and other geometry are maintained. In the **Part Navigator**, NX creates a new 2D component node under the sketch node of the sheet or the drawing view or under the parent 2D component node.

Where do I find it?

Application	Drafting
Command Finder	**Define 2D Component**
Graphics window	Right-click selection→**Define 2D Component**
Part Navigator	Right-click the 2D component node→**Define 2D Component**
Reuse Library	Right-click a folder node or a subfolder node under the **2D Component Library**→**Define 2D Component**

Replace 2D Component

Use the **Replace 2D Component** command to replace the selected 2D component instances with another 2D component definition.

You can replace:

- One 2D component instance.
- All instances of the selected 2D component.
- Instances of different 2D components.
- Nested 2D component instances. This causes the parent 2D component definitions to change and the 2D component instances that use them to update.

When the association between the 2D component instance and its definition is broken, the **Replace 2D Component** command can replace the orphan 2D component with an existing 2D component definition. The association can be broken, if, for example, the 2D component instance of a local 2D component is copied from one part to another without publishing the 2D component definition or if the part file of a published 2D component is deleted from the system drive or from Teamcenter.

You can replace locked 2D components as long as their parents are not locked. The replaced instances are not locked. You cannot replace an active 2D component. You must first exit or finish editing the 2D component before replacing it.

Where do I find it?

Application	Drafting
Prerequisite	The 2D component must not be active.
Command Finder	**Replace 2D Component**
Graphics window	Right-click the 2D component→**Replace**
Part Navigator	Right-click the 2D component node→**Replace**

Make 2D Component Unique

Use the **Make 2D Component Unique** command to create a new 2D component definition for the selected 2D component instance. In the **Make 2D Component Unique** dialog box, you specify:

- The location in the **Reuse Library** where you want to store the new definition.
- The descriptive name of the 2D component definition.
- The optional preview image.

When you create a new 2D component definition, the nested 2D components of the selected instance keep the same definitions they had prior to the command. Making a nested 2D component instance unique changes the definition of its parent and updates its instantiated components.

When the association between the 2D component instance and its definition is broken, the **Make 2D Component Unique** command creates a new 2D component definition for the orphan 2D component. The association can be broken, if, for example, the 2D component instance of a local 2D component is copied from one part to another without publishing the 2D component definition or if the part file of a published 2D component is deleted from the system drive or from Teamcenter.

You can make a locked 2D component unique as long as its parent is not locked. You cannot make an active 2D component unique. You must first exit or finish editing the 2D component before making it unique.

Where do I find it?

Application	Drafting
Prerequisite	The 2D component must not be active.
Command Finder	**Make 2D Component Unique**
Graphics window	Right-click the 2D component→**Make Unique**
Part Navigator	Right-click the 2D component node→**Make Unique**

Smash 2D Component

Use the **Smash 2D Component** command to break an instance of a 2D component into its constituent pieces. NX inserts the constituent pieces into the parent sketch.

If you smash a 2D component instance that has nested 2D components, the nested 2D components are not smashed, but they are placed in the top level sketch or the parent 2D component. You can smash locked 2D components as long as their parents are not locked.

Constraints, dimensions, and annotations applied to the smashed 2D component are maintained.

Instances are smashed the way they are currently being displayed. If the instance is out-of-date the smashed objects match the out-of-date instance. If the annotations, dimensions, and reference curves are not visible due to defined style options, they are copied but remain hidden in the smashed instance.

You cannot smash an active 2D component. You must first exit or finish editing the 2D component before smashing it.

Where do I find it?

Application	Drafting
Prerequisite	The 2D component must not be active.
Command Finder	**Smash 2D Component**
Graphics window	Right-click the 2D component→**Smash**
Part Navigator	Right-click the 2D component node→**Smash**

Publish 2D Component

Use the **Publish 2D Component** command to create an external part file containing the 2D component definition. You can publish one 2D component definition or all 2D component definitions that are stored in the **Local 2D Components** folder or its subfolders.

In the **Publish 2D Component** dialog box, you specify one destination location where all 2D component definitions will be saved. When you publish a 2D component definition that contains nested 2D components, NX also publishes the nested 2D component definitions in the same destination location.

In the **Customer Defaults** dialog box, you or your company can specify:

- The name and location of the published 2D component library.
- If the published 2D components are stored either on a disk drive or in Teamcenter.

In native NX, the external 2D component part files have distinct names that end with *.2dc.prt*. In Teamcenter Integration, the published 2D components are items with unique names across the Teamcenter database.

Where do I find it?

Application	Drafting
Reuse Library	Right-click the **Local 2D Components** folder node or a subfolder node→**Publish 2D Component** Right-click a 2D component in the **Local 2D Components** folder→**Publish**

Customer Defaults

Command Finder	**Customer Defaults**
Location in dialog box	**Gateway**→**Reuse Library**→**2D Component Library** tab

Create Assembly from Layout

Use the **Create Assembly from Layout** command to create an assembly that replicates the 2D component geometry and hierarchy of your layout. This enables you to continue your design in 3D in the Modeling application.

You specify:

- The top level sketch that contains the layout. The sketch can be either in the sheet or the drawing view.
- A part file where NX creates the assembly containing the component parts. The units of the assembly part must be the same as the units of the layout part.

NX creates:

- The assembly structure that replicates the 2D component hierarchy of your layout.

- A new part for each 2D component definition that is used in the selected layout. The units of the created component parts are the same as the units of the layout part. The name of the created component part file is the same as the name of the 2D component definition.

  Note

  If a file or an item with the same name already exists in the destination folder or in the Teamcenter database, NX does not create the part. Instead, you get a warning message.

The geometry of the 2D definition is used to create all component parts, even if the 2D component instance is locked or out-of-date. This may cause differences between the layout and the assembly if the layout contains out-of-date instances.

In the assembly, NX does not create parts from orphan 2D component instances as they do not have a definition. If you have orphan 2D components use the **Make 2D Component Unique** command to create 2D component definitions for them before creating the assembly.

Transferred annotations become assorted parts.

Sketches in Modeling

The layout top level sketch is copied to the assembly part sketch, while each 2D component definition sketch is copied to the corresponding component part sketch. The sketches include all geometry, geometric constraints, and dimensional constraints from the original sketches.

All sketches are created in the XY plane of their corresponding new part and their origin is placed at the origin of the absolute coordinate system. If the original top level sketch is in a drawing view, the drawing view orientation is used to define the plane. The sketch origin in a component part corresponds to the position of the 2D component anchor point. The XY orientation of the created sketch matches the XY orientation of the 2D component sketch. The display settings of the 2D component geometry is preserved in the component part sketch.

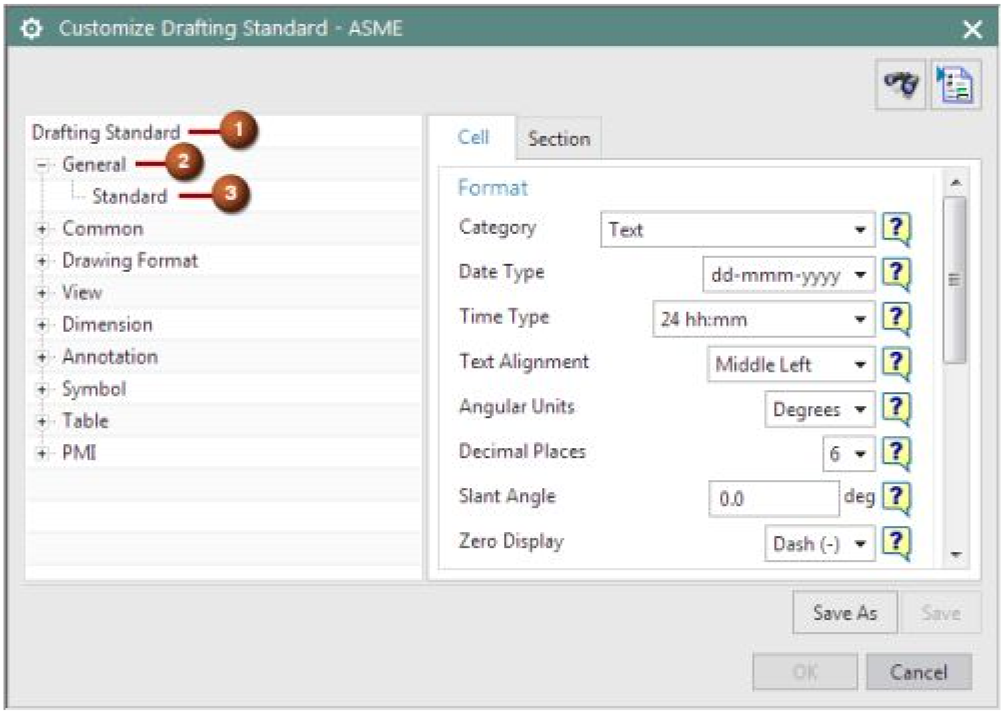
Where do I find it?

Application	Drafting
Command Finder	**Create Assembly from Layout** 

Restructure of the Drafting customer defaults

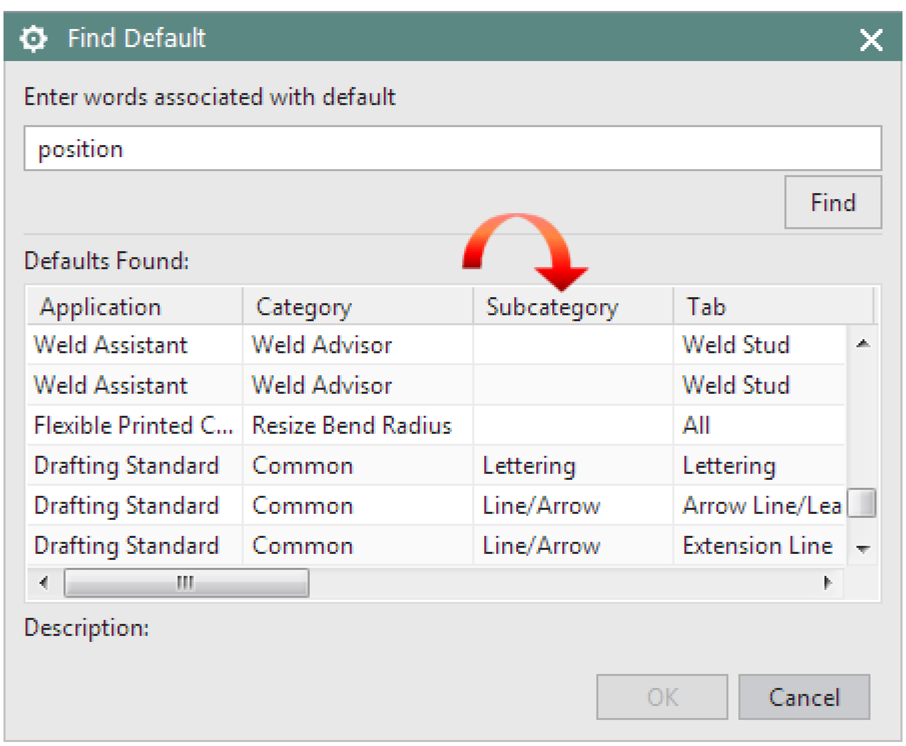
What is it?

The Drafting customer default categories and options are reorganized so that they are arranged similar to the way the same options are arranged in the **Drafting Preferences** dialog box. This reorganization introduces a new identifier in the structure of a customer defaults file called **Subcategory**. Subcategories group the defaults into smaller and more logical collections of options.



1 = Application
2 = Category
3 = Subcategory

The **Subcategory** identifier appears as an additional column in the results table when you search for a default.



Most drafting options are now located under the **Drafting Standard** customer default option. In the **Customer Defaults** dialog box, from the **Drafting→General/Setup** node, you must select an appropriate Drafting standard, and then click **Customize Standard** to access and change these options.

Additionally, you can now use the **Find Default** tool to search for a default like in the **Customize Standard** mode.

Drafting customer default files

Due to this reorganization, most drafting customer default settings are now located in, and controlled by, the appropriate drafting standards file. For example, if the **Drafting Standard** for your part is set to **ISO**, then most of the drafting custom default settings are located in the *nx_ISO_Drafting_Standard_Shipped.dpv* file.

> **Tip**
>
> If you have customized your Drafting default settings, or have created unique drafting standard default files (e.g. a custom version of the *nx_ISO_Drafting_Standard_Shipped.dpv* file), you must use the **Import Drafting Standard** tool to migrate all of your Drafting customer default settings into NX 10. To access this tool, you must click the **Customize Standard** button on the **Drafting→General/Setup** node in the **Customer Defaults** dialog box.
>
> If you have customized your customer defaults file (i.e. *nx9_user.dpv*) then you must use both the **Import Drafting Standard** tool and the **Import Defaults** tool, located in the **Manage Current Settings** dialog box, to import all of your customized user default settings.

Where do I find it?

Access and change existing Drafting customer default settings

Command Finder	**Customer Defaults**
Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Drafting→General/Setup** node→**Standard** tab→select a standard from the **Drafting Standard** list→click **Customize Standard**

Import your existing drafting customer defaults settings

Command Finder	**Customer Defaults**
Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Drafting→General/Setup** node→**Standard** tab→select a standard from the **Drafting Standard** list→click **Customize Standard** and then click the **Import Drafting Standard** button

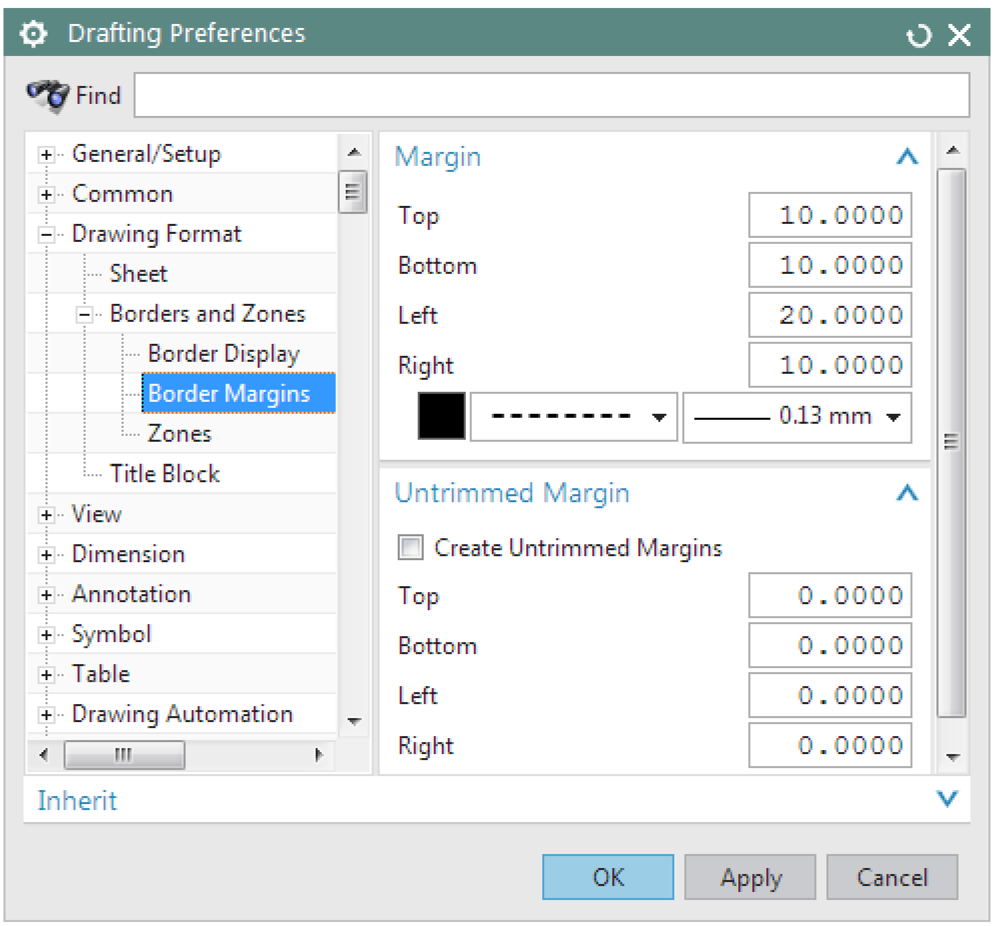
Borders and Zones enhancements

What is it?

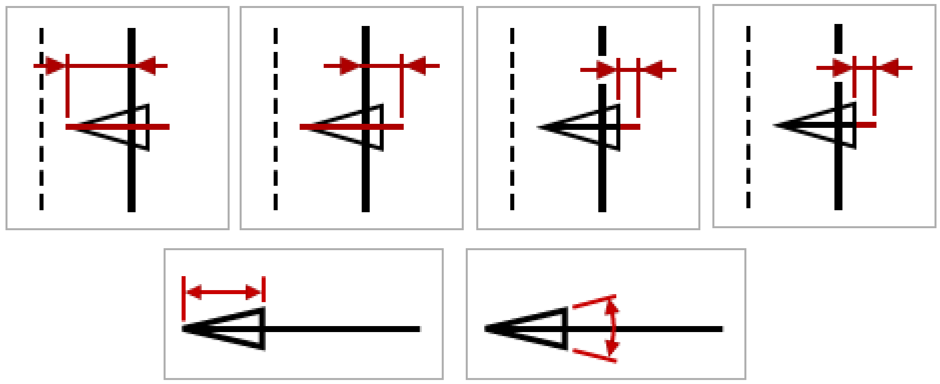
To provide improved compliance with drafting standards and improve the usability of the **Borders and Zones** command, the following enhancements are available:

- The **Borders and Zones** dialog box has been redesigned to provide a more intuitive interaction with the command.

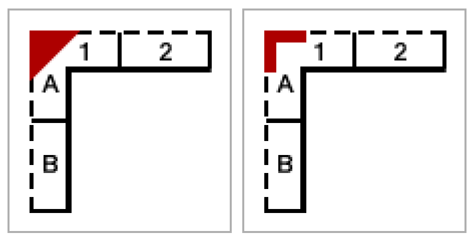
- Format and display options originally on the **Borders and Zones** dialog box are now accessible from a new **Borders and Zones** node in the **Drafting Preferences** dialog box and **Settings** dialog box.



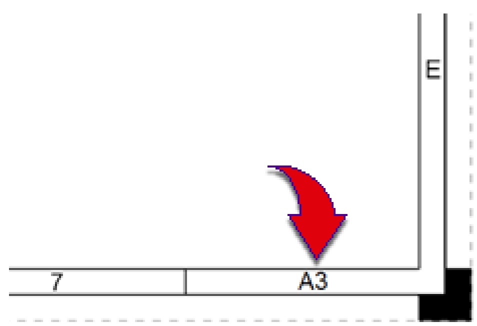
- New preference and setting options that let you do the following:
  - o Specify the drafting standard that is used to set the default values when creating borders and zones.
  - o Control the appearance and position of line center marks and arrow head center marks.



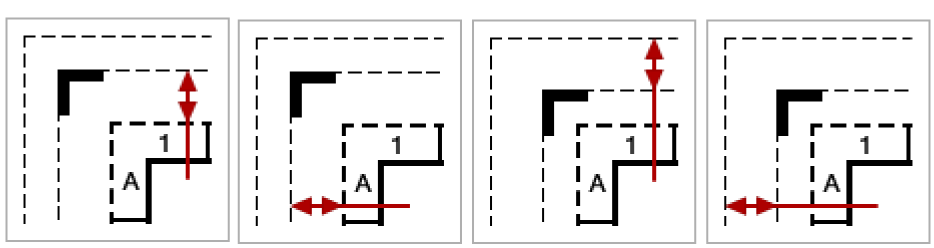
- Set the style and appearance of the trimming marks. You can choose to display a right-angled isosceles triangle trimming mark or a corner trimming mark.



- Display the drawing sheet size in place of the zone label in the lower right corner of the drawing border.



- Specify the letters that cannot be used in a zone label.
- Specify whether or not zone numbers can continue on subsequent drawing sheets.
- Set margin values used to locate the drawing border within trimmed and untrimmed drawing sheets.



Why should I use it?

Use the new preference settings to create more standards-compliant drawing formats, or to further refine your custom drawing format.

Where do I find it?

Borders and zones preferences

Application	Drafting
Command Finder	**Drafting Preferences**
Location in dialog box	**Drawing Format**→**Borders and Zones**→**Border Display** node **Drawing Format**→**Borders and Zones**→**Border Margin** node **Drawing Format**→**Borders and Zones**→**Zone** node

Borders and zones settings

Application	Drafting
Command Finder	**Borders and Zones**
Location in dialog box	**Settings** group→**Settings**

Title Block enhancements

What is it?

The following enhancements are available for title blocks:

- The title block automatically repositions correctly when the drawing border is changed or removed.
- A new **Edit Table** button in the **Define Title Block** dialog box places you in a special task environment with specific table commands for editing and redefining the table. This task environment also allows you to edit and delete locked title block cells. The cells revert to a locked state once the edits are complete.

  Note

  While in the task environment, you can add and remove sections, cells, columns and rows from any tabular note that makes up the title block table. You must exit the task environment and use the **Select Tables** button in the **Define Title Block** dialog box to add a new tabular note to, or completely remove an entire tabular note from, the title block table.

  To exit the task environment, you must select one of the following commands:

  - **Finish Edit Table** saves the table edits and exits the task environment.

  - o **Exit Edit Table** exits the task environment without saving any edits.

- The ability to cut, copy, and paste title blocks.

- A new **Date-Time** option for cells that lets you display both a date and time value, and control the format of those values.

  Note

  This option works correctly for title blocks and tabular notes created in NX 10 or higher. If you want to use this option in a title block or tabular note created before NX 10, you must use one of the version up options on the **General/Setup**→**Workflow** node in the **Drafting Preferences** dialog box to first update the title block or tabular note.

- Word wrapping is now supported for cells in title blocks.

  Note

  Applying any fit method to a title block table may cause the table to expand in an undesirable direction. Use care when applying fit methods to cells in a table.

- You can use the Enter key to cycle through the list of items in both the **Populate Title Block** dialog box and the **Define Title Block** dialog box.

Where do I find it?

Edit Table command

Application	Drafting
Prerequisite	A title block table must be selected.
Graphics window	Right-click a title block table→**Edit Definition**
Location in dialog box	**Tables** group→**Edit Table**

Finish Edit Table command and **Exit Edit Table** command

Application	Drafting
Prerequisite	You must be in the Edit Table task environment.
Command Finder	**Finish Edit Table** **Exit Edit Table**

Date-Time option for cells

Application	Drafting

Location in dialog box	In the **Drafting Preferences** dialog box: **Table**→**Common**→**Cell** node→**Format** group→**Category** list→**Date-Time** In the **Settings** dialog box: **Common**→**Cell** node→**Format** group→**Category** list→**Date-Time**

Fit support for angled text in tables

What is it?

The following fit methods are now supported for text at a 90° angle in the cell of a table:

- **Wrap**
- **Auto Size Text**
- **Truncate**
- **Overwrite Border**
- **Abbreviate**
- **Remove Spaces**
- **Auto Size Row**
- **Auto Size Column**

Why should I use it?

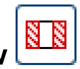
Use the fit methods when you want to display vertical text in the cell of a table while controlling the size of the cell.

Where do I find it?

Application	Drafting
Prerequisite	The **Lettering Angle** option in a cell must be set to something other than 0 degrees.
Location in dialog box	In the **Drafting Preferences** dialog box: **Table**→**Common**→**Cell** node→**Fit Methods** group In the **Settings** dialog box: **Common**→**Cell** node→**Fit Methods** group

Section View command

What is it?

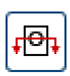
The following section view commands are consolidated into a single **Section View**  command.

- Simple/Stepped
- Half
- Revolved
- Folded
- Unfolded Point to Point

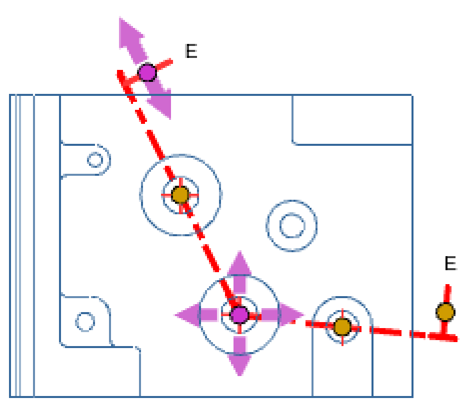
The section view dialog bar is replaced with a block-based dialog box that lets you select which type of section view you want to create, and then automatically displays the appropriate groups of options based on the view type and creation method you selected.

Section line definition

As with the previous commands, the **Section View** command requires you to define the section line before you create the section view. You can dynamically create the section line after selecting an existing drafting view, or you can select an existing stand-alone section line.

> **Note**
>
> Use the **Section Line**  command if you want to create a stand-alone section line using the sketch tools.

When you choose to dynamically create the section line, interactive on-screen controls are available to define and edit the location and configuration of the section line.



Example of on-screen control handles for end points and node points of a section line

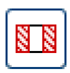
Legacy section views

The following section view types are not affected by this change, and continue to use the same dialog box interface and process for creating the view.

- Unfolded Point and Angle
- Pictorial
- Oriented
- Break-out

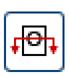
The **Unfolded Point and Angle** and **Oriented** section view commands are hidden in all roles and must be added to the user interface by using the **Customize** command on the Ribbon bar. The **Pictorial** and **Break-out** section view commands are available in the advanced roles, or they can be added to the user interface using the **Customize** command.

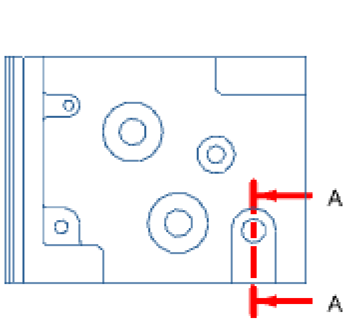
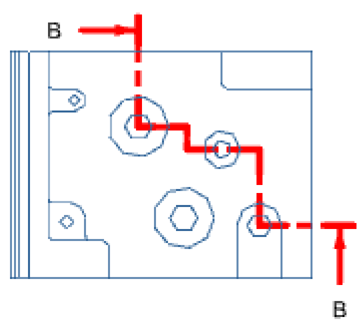
Where do I find it?

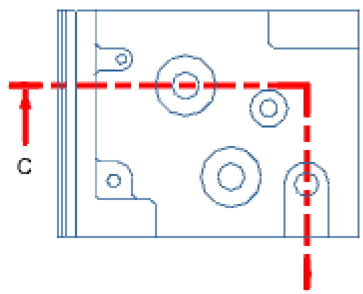
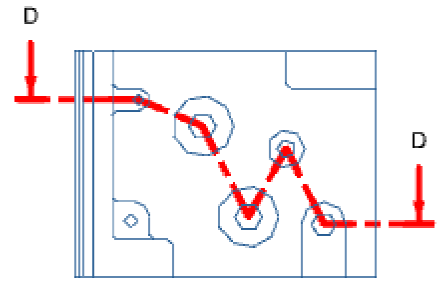
Application	Drafting
Command Finder	**Section View** 
Graphics window	Right-click a drafting view→**Add Section View** Right-click a stand-alone section line→**Add Section View**

Section Line command

What is it?

Section Line  lets you create standalone section lines for the following types of section lines.

 **Simple**	 **Stepped**

 **Half**	 **Point to point (folded and unfolded)**

The **Section Line** command always creates section lines in a sketch that has the following unique characteristics:

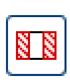
- Any dimensions created in the sketch are for reference only, and are not visible outside of the sketch environment.
- You can constrain the sketch curves to geometry in the associated view only. If you want to add constraints to geometry outside of the view, including geometry in other sketches associated to the view, you must add reference geometry to the sketch.
- Only one continuous section line can exist in the sketch.
- The sketch is not visible in the **Part Navigator**. You must edit the section line to enter the sketch environment.

When editing a section line, you cannot change the parent view. You cannot add sketch-based section lines to views that contain view breaks. If you want to add associative section lines to a drawing view with an **Exact (Pre-NX 8.5)** representation, you must turn on extracted edges.

Why should I use it?

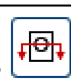
Use the **Section Line** command when you want greater control over the definition of the section line. You can also use the **Section Line** command to create a stand-alone section line that is not associated to a section view.

Note

Use the **Section View**  command if you want to simultaneously create both a section line and an associated section view.

Where do I find it?

Create a stand-alone section line

Application	Drafting
Command Finder	**Section Line** 
Graphics window	Right-click a drafting view→**Add Section Line**

Edit a section line

Application	Drafting
Graphics window	Right-click a section line→**Edit** Double-click a section line

View enhancements

What is it?

The following enhancements are available for smart lightweight drafting views:

- You can create break-out section views from smart lightweight views.
- Crosshatch patterns are correctly rendered when they are hidden behind bodies in a sectioned smart lightweight view.
- You can import PMI lightweight section views onto a drawing as smart lightweight section views.
- You can suppress the display of all hidden edges that are not referenced in a smart lightweight view. Alternately, you can display hidden edges that are referenced by associated annotation or when view dependent edits are applied to the hidden edge.

Also, the **Work in Progress** interruption mechanism has been enhanced to provide a quicker response when stopping a view update process.

Why should I use it?

Smart lightweight drafting views improve performance and memory efficiency when working with large assembly drawings. These enhancements enable a wider use of smart lightweight views in drawings.

Where do I find it?

Set the view type to display a smart lightweight view

Application	Drafting
Location in dialog box	In the **Drafting Preferences** dialog box: **View**→**Common** node→**Setting** group→**Representation** In the **Settings** dialog box: **Common** node→**Setting** group→**Representation**

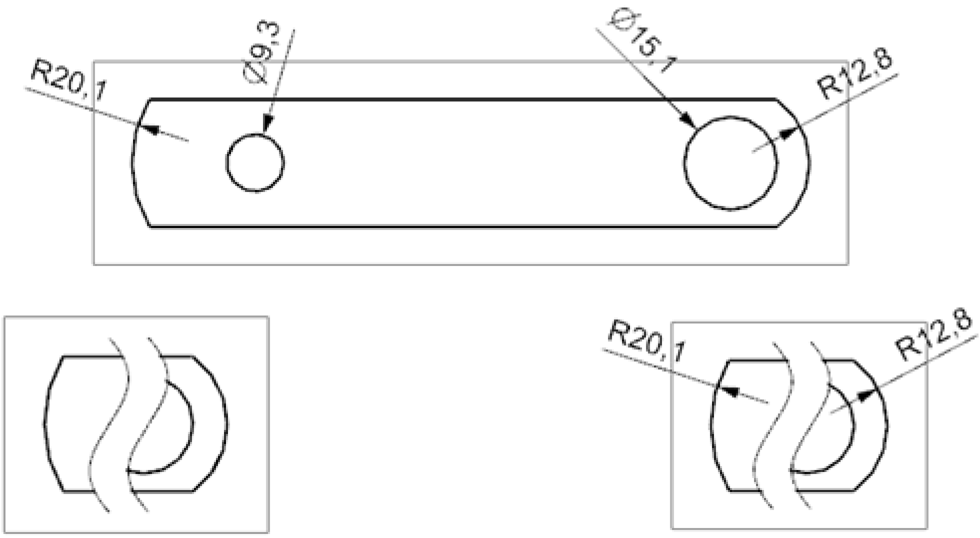
View break enhancements

What is it?

When you break a view that has a radial dimension partially outside of the view break, NX displays the radial dimension with the view break. The radial dimension is partially outside if its leader terminate point is outside of the view break. The selected curve for the radial dimension can be an arc, circle, spline, or conic.

This enhancement applies to any radial dimensions, whether their settings specify **Radius to Center** or not, with arrows pointing in or out. It applies both to drafting radial dimensions created on the drawing and to inherited PMI radial dimensions shown on the drawing.

In previous releases, the radial dimension was hidden when the center of the arc or circle was inside the view break.

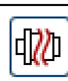


Pre-NX 10 **NX 10**

If you open a part with views created in older NX releases, you need to version the views up to NX 10 to display the radial dimension that is partially inside the view break.

If any dimension or annotation attachment point is fully consumed by a view break the object highlights so that you can see and acknowledge the result before it is hidden.

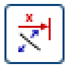
Where do I find it?

Application	Drafting
Command Finder	**View Break** 
Graphics window	Right-click a view border→**Add View Break**
Part Navigator	Right-click a view node→**Add View Break**

View versioning

Application	Drafting
Command Finder	**Drafting Preferences** 
Location in dialog box	**General/Setup**→**Workflow** node→**Version Up** group

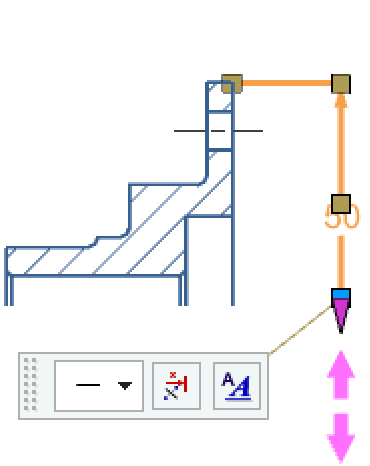
Single-sided dimensions

The **Display as Single Sided Dimension** option lets you create and edit linear and angular dimensions with only a single leader line. Use single-sided linear or angular dimensions for geometry that is larger than the size of the drawing view or sheet, or where the measurement origin is associated to geometry that is not visible. While creating or editing a single-sided dimension, you can use the **Flip Dimension Side**  button to switch between dimension lines.

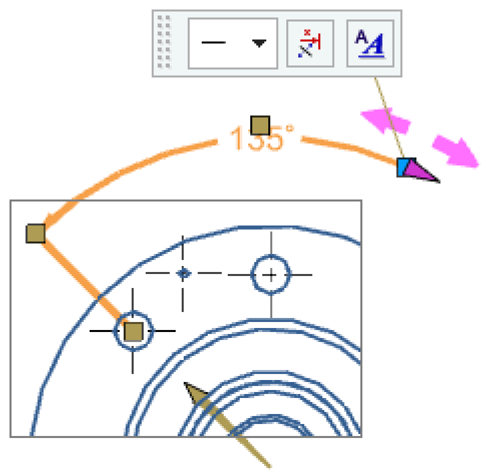
Note

Chain dimension sets and narrow dimensions are not supported. Also, you cannot create single-sided dimensions using the **Rapid** dimension command.

When you are in edit mode. Access handles are available to indicate which extension line and arrow to retain. The access handle can also be used to set the length of the remaining, stunted arrow line. Alternately, you can also set the length of the remaining line using the **Length** option in the **Settings** dialog box.



Example of a linear single–sided dimension



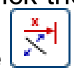
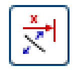
Example of an angular single–sided dimension

Where do I find it?

Set the single sided option

Application	Drafting and PMI
Graphics window	Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Display as Single Sided Dimension**

Flip Dimension Side button

Application	Drafting, PMI
Graphics window	While in edit mode, click the Arrow Line access handle, and then turn the **Flip Dimension Side**  on-screen button on or off. Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Flip Dimension Side** 

Set the length of the remaining arrow line

Application	Drafting, PMI
Graphics window	While in the edit mode, click and drag the Arrow Line access handle Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Length**

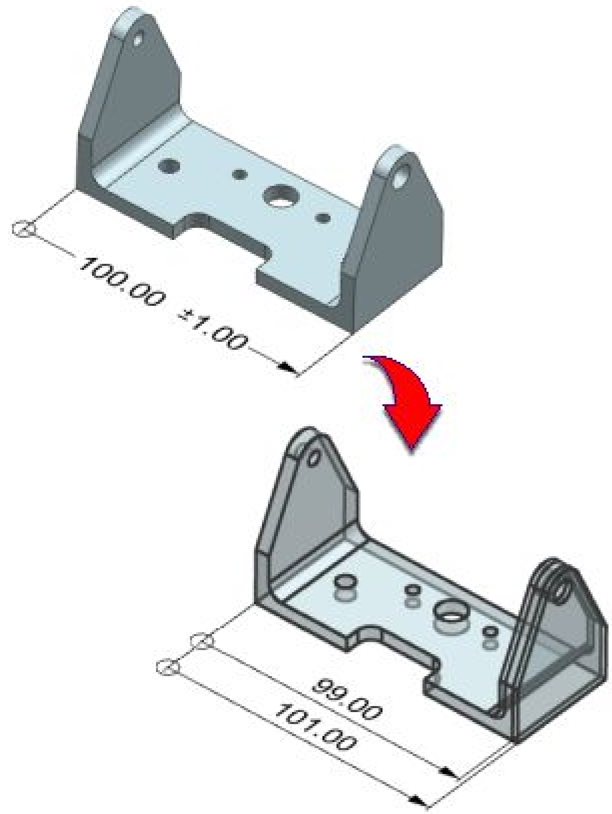
Linear dimension measurement types

What is it?

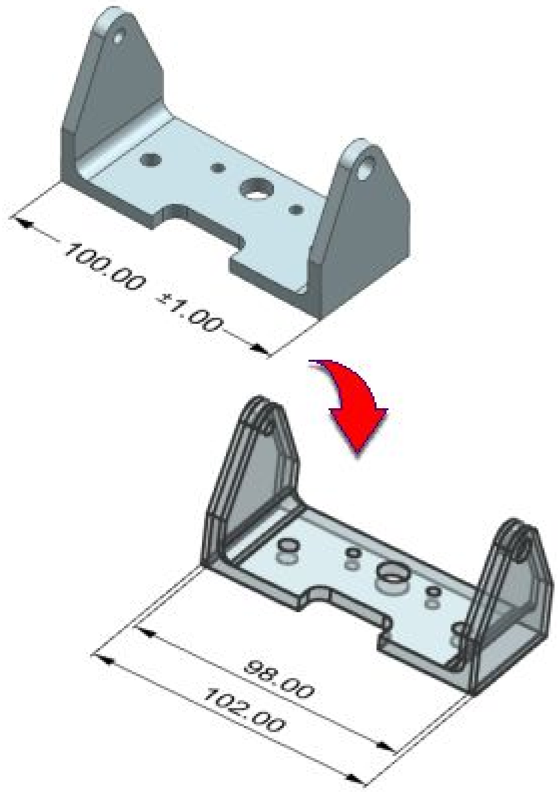
A new **Measurement Type** option in the **Linear Dimension** dialog box lets you specify a linear dimension as a general dimension, a directed dimension, or a feature of size dimension. The dimension type determines how the feature is measured and how the tolerance is applied to the measurement. This additional information is saved with the dimension object when it is created.

- A **General** dimension is a simple dimension with no additional, implied information about the measurement.

- A **Directed** dimension specifies the direction in which a feature is measured from an implied datum. All of the tolerance is applied to the second object.



- A **Feature of Size** dimension is a measurement about an implied datum halfway between the two measured features. The tolerance is equally disposed about the two measured features.



When the **Measurement Type** is set to **Directed**, you can select multiple objects for the origin object and the measured object. When it is set to **Feature of Size**, you can select multiple objects for the first object and the second object

Note

To make the **Measurement Type** option available in the **Linear** dimension dialog box, you must first set the **Enable Directed Dimensions** option in the **Drafting Preferences** dialog box. This option is not available in the **Settings** dialog box, so you cannot set it while creating or editing a linear dimension

Hole callouts, cylindrical dimensions, chain dimensions, and baseline dimensions are not supported as directed or feature of size dimensions.

Why should I use it?

Use the new measurement types to capture additional design information when dimensioning the features on your model. In particular, the information captured for PMI dimensions can be used in downstream applications such as tolerance analysis and inspection applications.

Where do I find it?

Measurement Type option

Application	Drafting, PMI

Prerequisite	The **Enable Directed Dimensions** option in the **Drafting Preferences** dialog box must be set.
Command Finder	**Linear**
Location in dialog box	**References** group→**Measurement Type**

Enable Directed Dimensions option

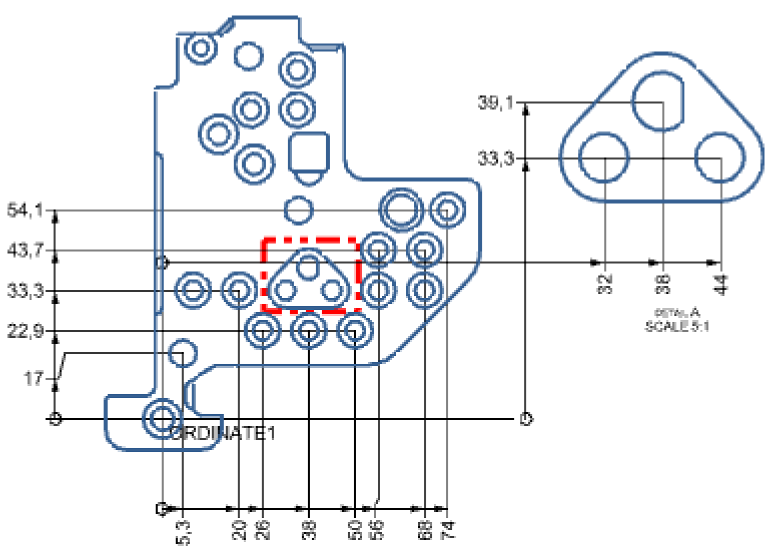
Application	Drafting, PMI
Command Finder	**Drafting Preferences**
Location in dialog box	**Dimensions**→**Workflow** node→**Enable Directed Dimensions**

Ordinate dimension enhancements

What is it?

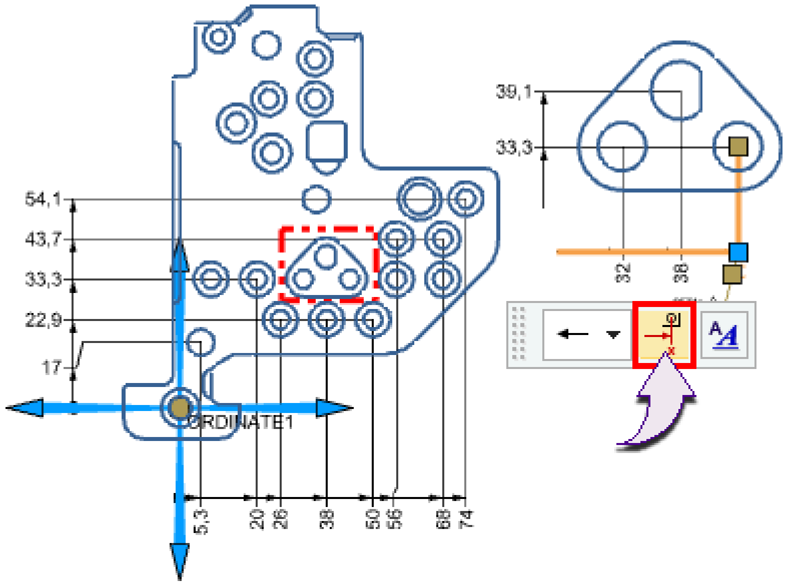
The following enhancements are available for ordinate dimensions:

- You can reuse existing ordinate origins in any view on any drawing sheet to create dimensions in other views.



  You can select an existing ordinate origin from a new **List** box in the **References** group of the **Ordinate Origin** dialog box, or you can select an existing ordinate origin or ordinate dimension directly from the graphics window.

- You can use the new **Clip Ordinate Dimension Line** on-screen setting to limit the ordinate dimension line to the bounding box of the view (not the actual view boundary).



This setting is only available when the ordinate origin and the ordinate dimensions are in different views or on different sheets and the **Show Dimension Line** option in the **Settings** dialog box for the ordinate dimension is set.

You can also set the **Clip Ordinate Dimension Line** option from the **Settings** dialog box while creating or editing an ordinate dimension.

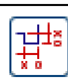
- You can locate and display the ordinate origin by right-clicking an existing ordinate dimension and selecting **Navigate to Origin**.

Why should I use it?

Use existing ordinate origins to create new ordinate dimensions in any view on any drawing sheet without the need to redefine the origin point. The **Navigate to Origin** command helps you locate the associative ordinate origin. The associative origin can be on the same drawing sheet or a different sheet.

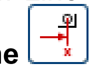
Where do I find it?

List of existing ordinates

Application	Drafting
Command Finder	**Ordinate** 
Location in dialog box	**References** group→**List** group

Clip Ordinate Dimension Line option

Application	Drafting

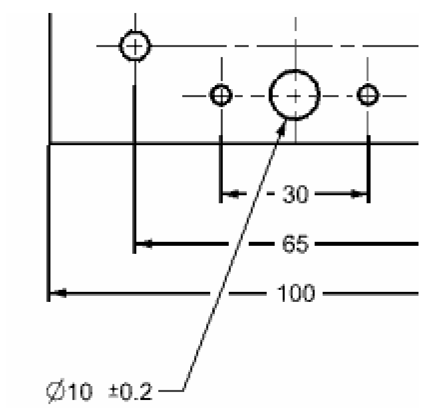
Prerequisite	In the **Settings** dialog box or **Drafting Preferences** dialog box, in the **Ordinate** node→**Set** group, the **Show Dimension Line** option must be set
Graphics window	While in the edit mode for an ordinate dimension, click the Arrow Line access handle, and then turn the **Clip Ordinate Dimension Line**  on-screen button on or off.
Location in dialog box	In the **Drafting Preferences** dialog box: **Common**→**Line/Arrow**→**Arrow Line** node→**Arrow Line** group→**Clip Ordinate Dimension Line** In the **Settings** dialog box: **Line/Arrow**→**Arrow Line** node→**Arrow Line** group→**Clip Ordinate Dimension Line**

Navigate to Origin command

Application	Drafting
Graphics window	Right-click an ordinate dimension→**Navigate to Origin**

Automatic line breaks

The **Create Break** option for dimension lines and annotation lines automatically creates breaks on the lines when they cross other annotation lines or dimension lines. The breaks appear and are centered at the intersection points once the annotation or dimension is placed.



A **Break Size** option lets you control the size of the break gap.

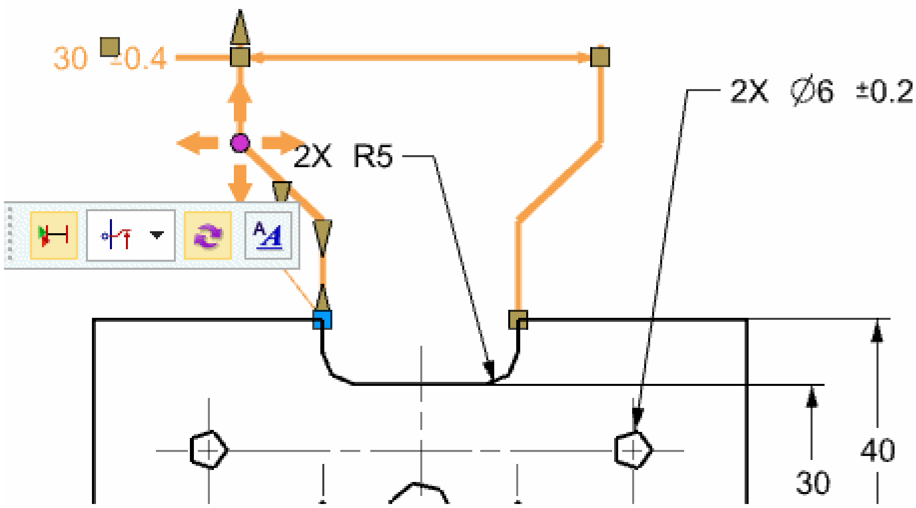
Where do I find it?

Application	Drafting

Location in dialog box	In the **Drafting Preferences** dialog box: **Common**→**Line/Arrow**→**Break** node→**Format** group→**Create Breaks** and **Break Size** In the **Settings** dialog box: **Line/Arrow**→**Break** node→**Format** group→**Create Breaks** and **Break Size**

Jogs on linear dimensions

You can add or remove jogs on extension line while creating or editing linear dimensions. Interactive on-screen handles and controls let you add, place, and size the jog for one extension line or for both extension lines.



Preference and setting options are available to control the initial parameters of the jog.

Where do I find it?

Interactively add or remove jogs

Application	Drafting and PMI
Graphics window	Double-click an existing dimension, or while creating a dimension, click the **Edit** on-screen button to enter the dimension edit mode, and then click the Extension Line access handle to display the on-screen controls.

Set the jog parameters

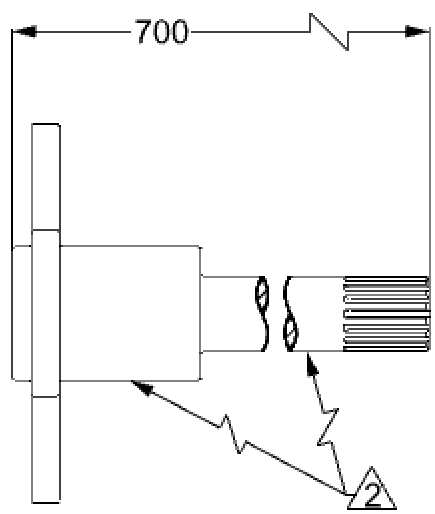
Application	Drafting and PMI

Location in dialog box	In the **Drafting Preferences** dialog box: **Dimension**→**Jog** node→**Linear Format Side 1** group and **Linear Format Side 2** group In the **Settings** dialog box: **Jog** node→**Linear Format Side 1** group and **Linear Format Side 2** group

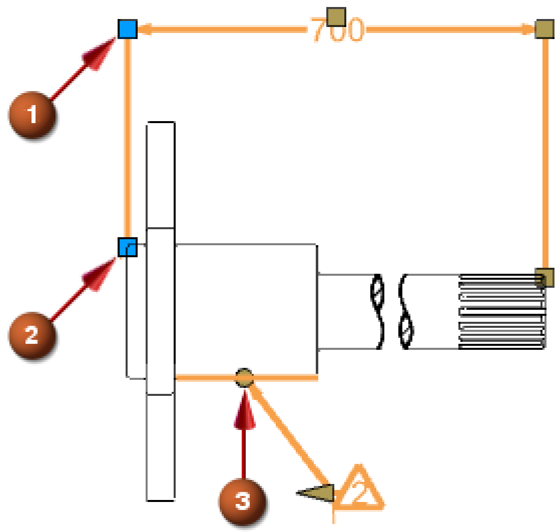
Foreshortening symbols

The following commands are available for adding and removing foreshortening symbols from leader lines and dimension lines:

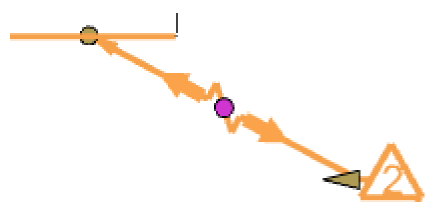
- **Add Foreshortening Symbol** lets you add a foreshortened symbol to leader lines and dimension extension lines.
- **Remove** lets you remove a single foreshortening symbol.
- **Remove all Foreshortening Symbols** lets you remove all foreshortening symbols associated with the dimension or annotation.



The commands for adding and removing foreshortened symbols are on a shortcut menu which is available while creating or editing annotations and dimensions. You display the shortcut menu by right-clicking an Arrow Line access handle (1), a Dimension Line access handle (2), or a Leader Line access handle (3).



Once the foreshortening symbol is created, you can use the drag handle to interactively reposition the symbol or delete the symbol.



Use options in the **Drafting Preferences** dialog box and **Settings** dialog box to control the type, width, height, and angle of the foreshortening symbol.

Where do I find it?

Add or remove foreshortening symbols

Application	Drafting
Graphics window	Double-click an existing dimension, or while creating a dimension, click the **Edit**  on-screen button to enter the dimension edit mode, and then do one of the following: • Right-click the Extension Line, Arrow Line, or Leader Line access handle→**Add Foreshortening Symbol** or **Remove All Foreshortening Symbols** • Right-click the drag handle of an existing foreshortening symbol→**Remove**

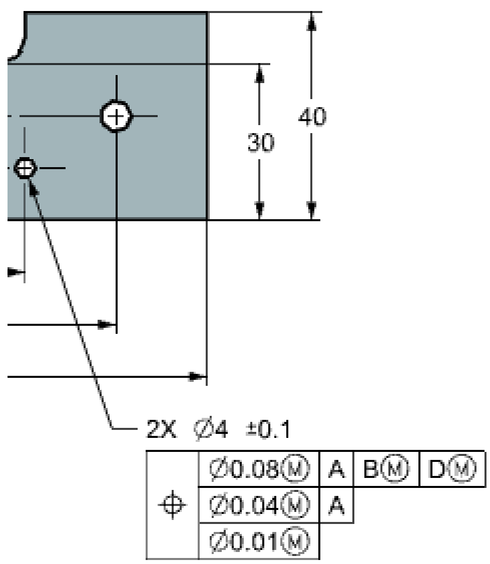
Foreshortening symbol options

Application	Drafting
Location in dialog box	In the **Drafting Preferences** dialog box: **Common→Line/Arrow→Foreshortening Symbol** node→**Format** group In the **Settings** dialog box: **Line/Arrow→Foreshortening Symbol** node→**Format** group

Adding segments to a composite Feature Control Frame

What is it?

You can use the new **Add New Frame** option in the **Feature Control Frame** dialog box to add segments to a composite feature control frame.



Why should I use it?

When creating tolerance information for a feature, you may require multiple tolerance segments to fully specify the location and definition of the feature.

Where do I find it?

Application	Drafting and PMI
Command Finder	**Feature Control Frame**
Prerequisite	In the **Characteristics** group, set the **Frame Style** option to **Composite Frame**.

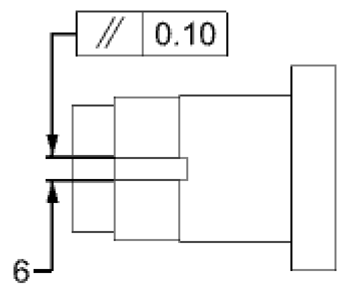
Location in dialog box	**Characteristics** group→**Add New Frame**

Leader enhancements and feature control frame display enhancements

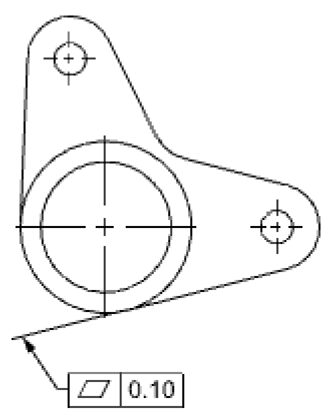
What is it?

Enhancements to leader lines and feature control frame functionality let you do the following:

- Use a new **Lock Perpendicular** shortcut menu option to lock a leader line so that it is always perpendicular to its attachment object. This option is only available for leader lines with a **Plain**, **All Around**, **Without Stub**, or **All Over** leader type.



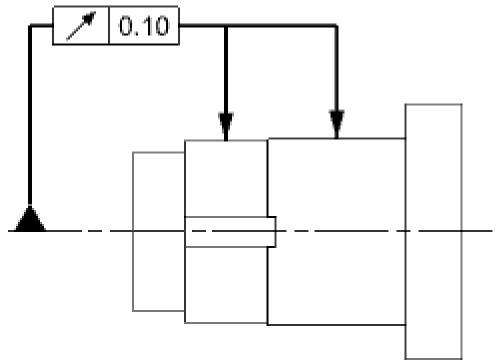
- Hold the Shift key and drag to create a leader with an extension line. This gesture is only available for annotation that has a single leader line with a **Plain**, **All Around**, **Without Stub**, or **All Over** leader type.



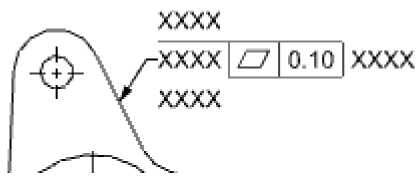
Note

When an extension line is added to a leader for a feature control frame, you cannot add additional leaders to the feature control frame.

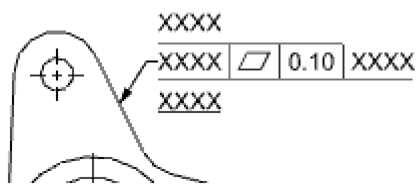
- Add multiple leader lines to single and compound feature control frames with a datum leader. Only **Plain**, **All Around**, and **All Over** leader types can be added.



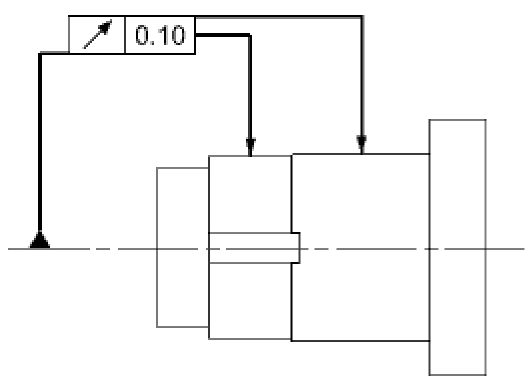
- Underline the text around a feature control frame. You can do the following:
    - o Underline only the first line of text around a feature control frame.



- o Underline all of the text around a feature control frame.



- Control the vertical attachment point on a feature control frame for all leader types.



Why should I use it?

These enhancements let you create standards-compliant annotation and feature control frame callouts.

Where do I find it?

Lock Perpendicular option

Application	Drafting and PMI
Prerequisite	Only one leader can be selected at a time, and the leader type must be set to **Plain**, **All Around**, **Without Stub**, or **All Over**
Graphics window	After selecting the leader termination object, right-click→**Lock Perpendicular**

Leader extension line

Application	Drafting and PMI
Prerequisite	The annotation object must have only one leader and the leader type must be set to **Plain**, **All Around**, **Without Stub**, or **All Over**
Graphics window	Before placing the annotation, hold the Shift key and drag

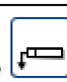
Multiple leader lines

Application	Drafting and PMI
Prerequisite	Only one datum leader can exist for the feature control frame.
Graphics window	Before placing a feature control frame, right-click→**Add New Leader**

Underline additional text

Application	Drafting and PMI
Location in dialog box	In the **Drafting Preferences** dialog box: **Annotation**→**GDT** node→**Feature Control Frame** group→**Underline Additional Text** In the **Settings** dialog box: **GDT** node→**Feature Control Frame** group→**Underline Additional Text**

Vertical attachment point

Application	Drafting and PMI
Command Finder	**Feature Control Frame**  
Location in dialog box	**Leader** group→**Style** group→**Vertical Attachment**

Adding symbols and text to datum references

What is it?

You can now add text or symbols to a datum reference within a feature control frame.

This enhancement adds a **Text** group for text and symbols to the **Compound Datum Reference** dialog box and an option for a **Projected** modifier to the **Feature Control Frame** and **Compound Datum Reference** dialog boxes.

When a feature control frame is selected, any extended information for datum references is displayed in the **Details** pane of the **Part Navigator**.

Related to the enhancement, the PMI search **Feature Control Frame Search Details** dialog box now shows only one datum reference at a time when setting search criteria. In order to limit a PMI search to feature control frames with references to two or more datums, you now create more than one feature control frame detail entry and add it to the list of PMI search criteria. If a saved PMI search model view in a legacy part contains feature control frame criteria with multiple datum reference fields enabled, when you edit the view, NX converts the criteria to the new format.

Columns for the new datum reference extended text are added to the output of a **PMI Report** command.

This extended qualifying information enhancement relates to the following standards sections:

- ASME Y14.5-2009

  Datum Translation Symbol
  Clarifying Applicable Maximum Material Boundary
  Multiple Least Material Boundaries
  Customized Datum Reference Frames

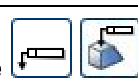
- ISO 5459:2011

  Datum modifier symbols

Why should I use it?

You can indicate conditions for how the datum is applied.

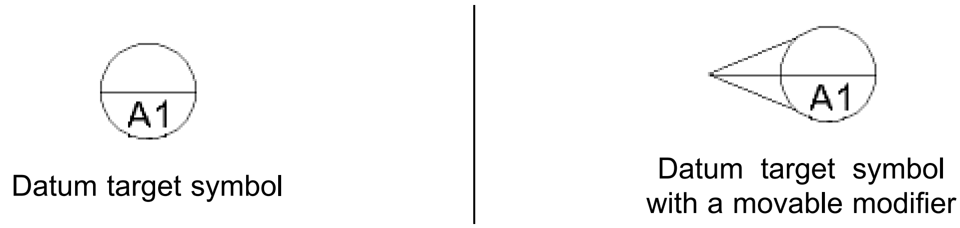
Where do I find it?

Application	Drafting and PMI
Command Finder	**Feature Control Frame** 
Location in dialog box	**Frame** group→**Primary Datum Reference**, **Secondary Datum Reference**, or **Tertiary Datum Reference**→**Text** group

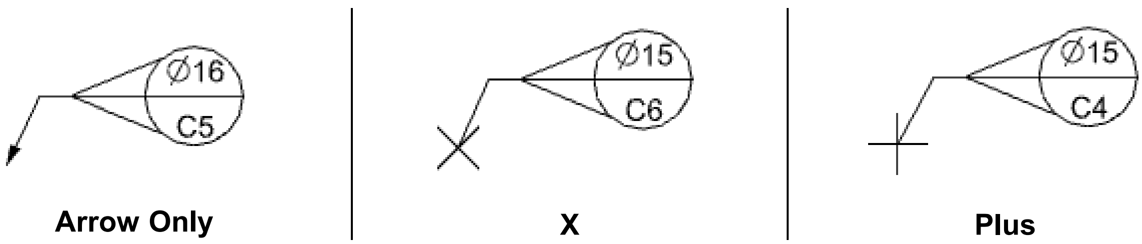
Movable datum targets, new terminator types

What is it?

A new **Movable** option lets you add a movable modifier to a datum target symbol.



Additionally, you can now specify the following leader termination types for datum targets.



The display of the movable modifier symbol is compliant with either the ASME or ISO drafting standard. When a standard other than ASME or ISO is in use, the display conforms to the ISO drafting standard. The orientation of the movable modifier may be adjusted when the ISO display is used.

Why should I use it?

Use this type of datum target when the form of the feature is such that an acceptable datum target can be anywhere along a location range. For example, round parts such as cast and forged parts that are fixtured using equalizing datums.

Where do I find it?

Application	Drafting and PMI
Command Finder	**Datum Target** 
Location in dialog box	**Target** group→**Movable** option and **Terminator** list

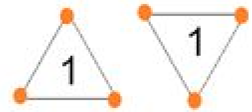
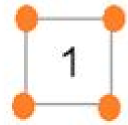
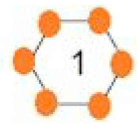
Balloon enhancement

What is it?

The new **Create Leader Attachment to Vertex** option, when set, automatically attaches leader lines without stubs to the vertex of a balloon symbol.

This option is available for the following balloon symbols:

Balloon Type	Available Vertices

Triangle Down, Triangle Up	
Square, Divided Square	
Hexagon, Divided Hexagon	

The vertex nearest to the terminating point of the leader line, or nearest to the last jog in the leader line, is used.

Note

By default, this option is set when the **Drafting Standard** for your current NX session is set to **ESKD**.

Where do I find it?

Application	Drafting
Prerequisite	The leader type for the balloon symbol must be set to **Without Stub**.
Location in dialog box	In the **Drafting Preferences** dialog box: **Annotation**→**Balloon** node→**Format** group→**Create Leader Attachment to Vertex** In the **Settings** dialog box: **Balloon** node→**Format** group→**Create Leader Attachment to Vertex**

Editing property tracking for inherited PMI

What is it?

Use the new **Inherit PMI Tracking** command to interactively change the status of tracked properties of inherited PMI in drawings. Tracked properties are updated in the drawing view when changes are made to the 3D PMI objects.

For example, from the **Inherit PMI Tracking** dialog box, you can enable the **AnnotationLocation** option to keep track of the position changes for all PMI annotation inherited onto the drawing.

If changes to NX result in new PMI properties that are not currently on the property tracking list, you can use the **Update Tracking Properties** option to add these new properties to the list.

Why should I use it?

Use this command to add tracked properties that are saved with the part file, or to modify the tracking status of existing tracked properties of inherited PMI objects on a drawing.

Where do I find it?

Change tracking status with the **Inherit PMI Tracking** command

Application	Drafting
Menu	**Format→PMI→Inherit PMI Tracking**

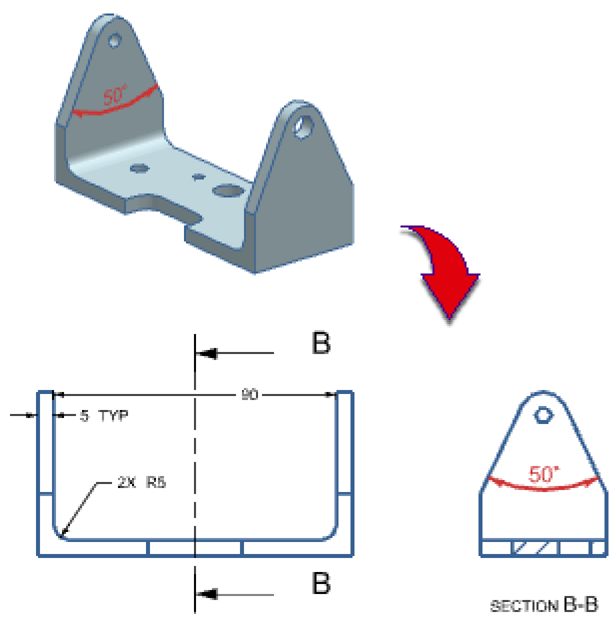
Add newly trackable properties with the **Update Tracking Properties** option

Application	Drafting
Prerequisite	Open a part that does not include all tracking properties supported by the current version of NX in its set of properties available for tracking.
Menu	**Format→PMI→Inherit PMI Tracking**
Dialog box	**Version Up** group→**Update Tracking Properties**

Inheriting PMI into a Drafting section view

What is it?

You can now use inherit PMI options to add PMI from a 3D model into a drafting section view.



You can add PMI to the following drafting section views:

Simple
Stepped
Half
Pictorial
Half Pictorial

All other section views are not supported.

The inherit PMI options set for the parent view are applied when you create the section view.

Note

For existing drafting section views in parts created in earlier versions of NX, you must version up the views before the inherit PMI options are available.

(**Drafting Preferences** dialog box→**General/Setup**→**Workflow**→**Version Up** group)

Why should I use it?

You can inherit PMI in a drafting section view instead of creating a PMI lightweight section view and then adding that view to your drawing. This approach lets you take advantage of reusing existing information and of bi-directionally editing your annotations.

Where do I find it?

Application	Drafting
Graphics window	Right-click the border of an existing Drafting section view→**Settings** →**Common**→**PMI**

API function for Drawing Booklets

What is it?

A new NX Open API, `Drafting::DrawingAutomationWizard`, lets you create and edit drawing booklets in a batch mode. Using this routine in conjunction with a User Exit allows you to suspend starting the booklet functionality until you reach a predefined time and place, or once you terminate the current NX session.

The User Exit registers a callback via the *JA_DRAWING_AUTOMATION_WIZARD_set_continue_processing* routine and, when the **Finish** button is selected in the **Drawing Booklet** wizard, the callback is executed.

Note

This callback mechanism is only supported in managed mode.

An interruption routine (*JA_DRAWING_AUTOMATION_WIZARD_set_continue_processing*) is also provided that lets you suspend the process before it is started. This interruption routine lets you control whether or not booklet drawings will be created after control returns to NX from the callback. If set to `TRUE`, the booklet will get created immediately. If set to `FALSE`, no drawings are created, and you can use the data returned by the callback to subsequently create the drawings themselves.

Why should I use it?

Processing drawing booklets consumes computer memory. Interactively running the **Drawing Booklet** wizard can consume your NX session and prevent you from interacting with and modifying drawings that have already been processed. You can use this API to create a custom program that will run the **Drawing Booklet** functionality at a predefined time and location, or when you terminate your NX session. This allows you to better manage your NX resources.

Multiple overlapping template regions

What is it?

Template regions are rectangular regions on a drawing template that allow the placement of only certain types of drafting objects when the template is used to create a drawing. The following enhancements to the **Template Region** command let you create more complex template regions.

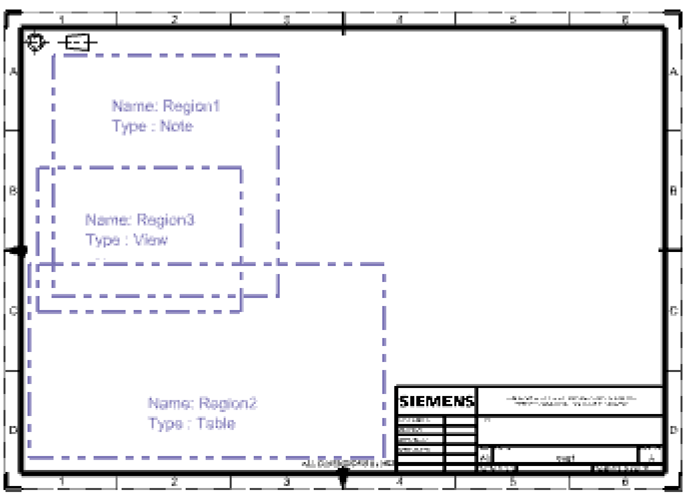
- You can now define more than two overlapping regions in a drawing template.

  Note

  Overlapping regions must have their growth option set in the same or opposite direction if you want the overflow to extend into the adjacent region.

- When the **Continuation** option to set to **Next Region**, any unlinked region of the same type, on the current sheet or any other drawing sheet, can be specified as the continuation region.

- The **Priority** setting for a region is now displayed when you choose to display template region information on the drawing sheet.



Why should I use it?

Use multiple overlapping template regions to create more complex drawing templates. Drawing templates are used by the **Drawing Booklet** wizard to automate the generation of a multi-sheet drawing.

Where do I find it?

Application	Drafting
Prerequisite	You must be in a drawing template, which was created using the **Mark as Template** command.

Command Finder	**Template Region** 

Drawing booklet enhancement

What is it?

You can now create or edit drawing booklets with no primary content. These booklets contain empty drawing sheets with no associated models.

- In native NX and in Teamcenter Integration, this will create an empty drawing with no references to components.
- In 4GD, this will create an empty drawing with no references to subsets.

To create an empty drawing , use the **Create Drawing Booklet** wizard but do not select any objects for the **Primary Content** option.

Why should I use it?

You can create empty drawings and then edit the drawings to customize the contents displayed on the sheets. For example, some drawing sheets can show components, and other sheets can contain related notes.

Where do I find it?

Application	Any application except Drafting
Prerequisite	You must be in an assembly file
Command Finder	**Create Drawing Booklet**
Menu	**Tools**→**Drawing Automation**→**Create Drawing Booklet** or **Edit Drawing Booklet**
Location in dialog box	**Scope** page→**Primary Content** group→**Select Object**

Specifying Item Type for custom symbols

What is it?

In a Teamcenter Integration environment, new options in the **Define Custom Symbol** dialog box and the **Define Symbol from Catalog** dialog box let you specify an **Item Type** in addition to specifying the name, revision, and number for the symbol. You can select the item type from a list of types that are available in the database. This list is typically the same as the list you see when the **Blank** template is selected in the **File New** dialog box.

Where do I find it?

Application	Drafting and PMI
Prerequisite	You must be in a Teamcenter Integration environment

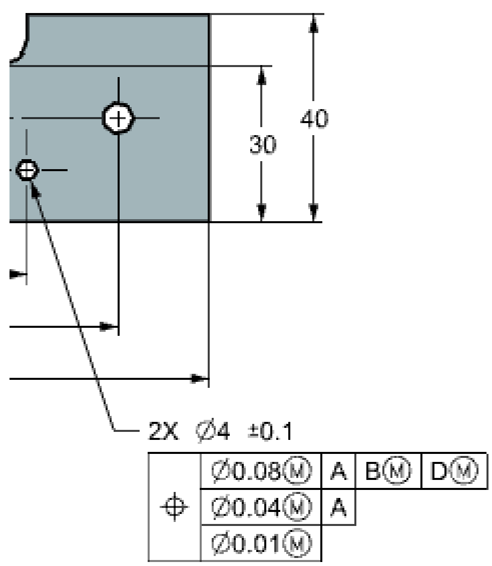
Command Finder	**Define Custom Symbol** **Define Symbol from Catalog**
Location in dialog box	**Symbol Name** group→**Item Type**

Product and Manufacturing Information (PMI)

Adding segments to a composite Feature Control Frame

What is it?

You can use the new **Add New Frame** option in the **Feature Control Frame** dialog box to add segments to a composite feature control frame.



Why should I use it?

When creating tolerance information for a feature, you may require multiple tolerance segments to fully specify the location and definition of the feature.

Where do I find it?

Application	Drafting and PMI
Command Finder	**Feature Control Frame**
Prerequisite	In the **Characteristics** group, set the **Frame Style** option to **Composite Frame**.

Location in dialog box	**Characteristics** group→**Add New Frame**

Adding symbols and text to datum references

What is it?

You can now add text or symbols to a datum reference within a feature control frame.

This enhancement adds a **Text** group for text and symbols to the **Compound Datum Reference** dialog box and an option for a **Projected** modifier to the **Feature Control Frame** and **Compound Datum Reference** dialog boxes.

When a feature control frame is selected, any extended information for datum references is displayed in the **Details** pane of the **Part Navigator**.

Related to the enhancement, the PMI search **Feature Control Frame Search Details** dialog box now shows only one datum reference at a time when setting search criteria. In order to limit a PMI search to feature control frames with references to two or more datums, you now create more than one feature control frame detail entry and add it to the list of PMI search criteria. If a saved PMI search model view in a legacy part contains feature control frame criteria with multiple datum reference fields enabled, when you edit the view, NX converts the criteria to the new format.

Columns for the new datum reference extended text are added to the output of a **PMI Report** command.

This extended qualifying information enhancement relates to the following standards sections:

- ASME Y14.5-2009

  Datum Translation Symbol
  Clarifying Applicable Maximum Material Boundary
  Multiple Least Material Boundaries
  Customized Datum Reference Frames

- ISO 5459:2011

  Datum modifier symbols

Why should I use it?

You can indicate conditions for how the datum is applied.

Where do I find it?

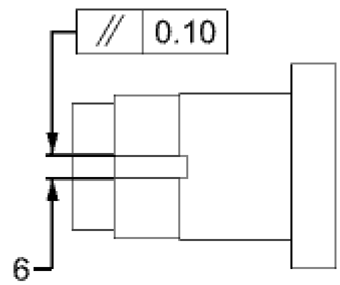
Application	Drafting and PMI
Command Finder	**Feature Control Frame**
Location in dialog box	**Frame** group→**Primary Datum Reference**, **Secondary Datum Reference**, or **Tertiary Datum Reference**→**Text** group

Leader enhancements and feature control frame display enhancements

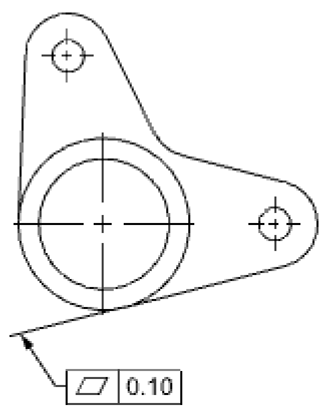
What is it?

Enhancements to leader lines and feature control frame functionality let you do the following:

- Use a new **Lock Perpendicular** shortcut menu option to lock a leader line so that it is always perpendicular to its attachment object. This option is only available for leader lines with a **Plain**, **All Around**, **Without Stub**, or **All Over** leader type.



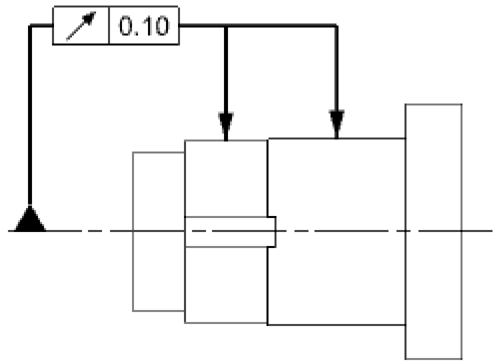
- Hold the Shift key and drag to create a leader with an extension line. This gesture is only available for annotation that has a single leader line with a **Plain**, **All Around**, **Without Stub**, or **All Over** leader type.



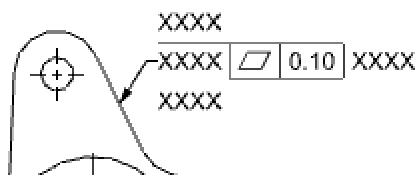
Note

When an extension line is added to a leader for a feature control frame, you cannot add additional leaders to the feature control frame.

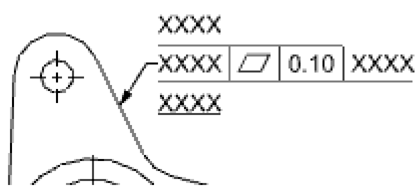
- Add multiple leader lines to single and compound feature control frames with a datum leader. Only **Plain**, **All Around**, and **All Over** leader types can be added.



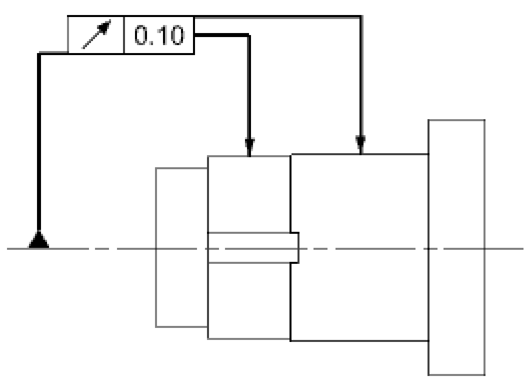
- Underline the text around a feature control frame. You can do the following:
  - Underline only the first line of text around a feature control frame.



  - Underline all of the text around a feature control frame.



- Control the vertical attachment point on a feature control frame for all leader types.



Why should I use it?

These enhancements let you create standards-compliant annotation and feature control frame callouts.

Where do I find it?

Lock Perpendicular option

Application	Drafting and PMI
Prerequisite	Only one leader can be selected at a time, and the leader type must be set to **Plain**, **All Around**, **Without Stub**, or **All Over**
Graphics window	After selecting the leader termination object, right-click→**Lock Perpendicular**

Leader extension line

Application	Drafting and PMI
Prerequisite	The annotation object must have only one leader and the leader type must be set to **Plain**, **All Around**, **Without Stub**, or **All Over**
Graphics window	Before placing the annotation, hold the Shift key and drag

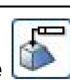
Multiple leader lines

Application	Drafting and PMI
Prerequisite	Only one datum leader can exist for the feature control frame.
Graphics window	Before placing a feature control frame, right-click→**Add New Leader**

Underline additional text

Application	Drafting and PMI
Location in dialog box	In the **Drafting Preferences** dialog box: **Annotation**→**GDT** node→**Feature Control Frame** group→**Underline Additional Text** In the **Settings** dialog box: **GDT** node→**Feature Control Frame** group→**Underline Additional Text**

Vertical attachment point

Application	Drafting and PMI
Command Finder	**Feature Control Frame** 
Location in dialog box	**Leader** group→**Style** group→**Vertical Attachment**

Enable Resize preference

What is it?

Use the new **Enable Resize** PMI preference to control whether or not the display size of PMI objects can change depending on the view scale.

When the **Enable Resize** preference is turned on, the following are available:

- **Resize on Create** (PMI preference)
- **Resize on View Save** (PMI preference)
- **Resize PMI** command

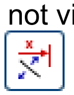
Why should I use it?

You can control availability of PMI object resizing methods.

Where do I find it?

Application	PMI
Command Finder	**PMI Preferences**
Location in dialog box	**Display** tab→**Resize Options** group→**Enable Resize**

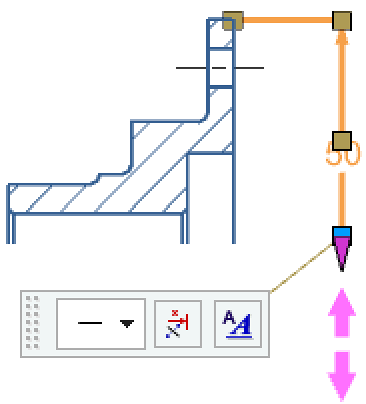
Single-sided dimensions

The **Display as Single Sided Dimension** option lets you create and edit linear and angular dimensions with only a single leader line. Use single-sided linear or angular dimensions for geometry that is larger than the size of the drawing view or sheet, or where the measurement origin is associated to geometry that is not visible. While creating or editing a single-sided dimension, you can use the **Flip Dimension Side**  button to switch between dimension lines.

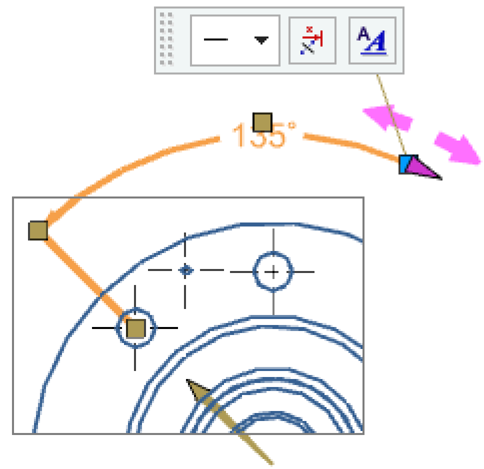
Note

Chain dimension sets and narrow dimensions are not supported. Also, you cannot create single-sided dimensions using the **Rapid** dimension command.

When you are in edit mode. Access handles are available to indicate which extension line and arrow to retain. The access handle can also be used to set the length of the remaining, stunted arrow line. Alternately, you can also set the length of the remaining line using the **Length** option in the **Settings** dialog box.



Example of a linear single–sided dimension



Example of an angular single–sided dimension

Where do I find it?

Set the single sided option

Application	Drafting and PMI
Graphics window	Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Display as Single Sided Dimension**

Flip Dimension Side button

Application	Drafting, PMI
Graphics window	While in edit mode, click the Arrow Line access handle, and then turn the **Flip Dimension Side** on-screen button on or off. Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Flip Dimension Side**

Set the length of the remaining arrow line

Application	Drafting, PMI

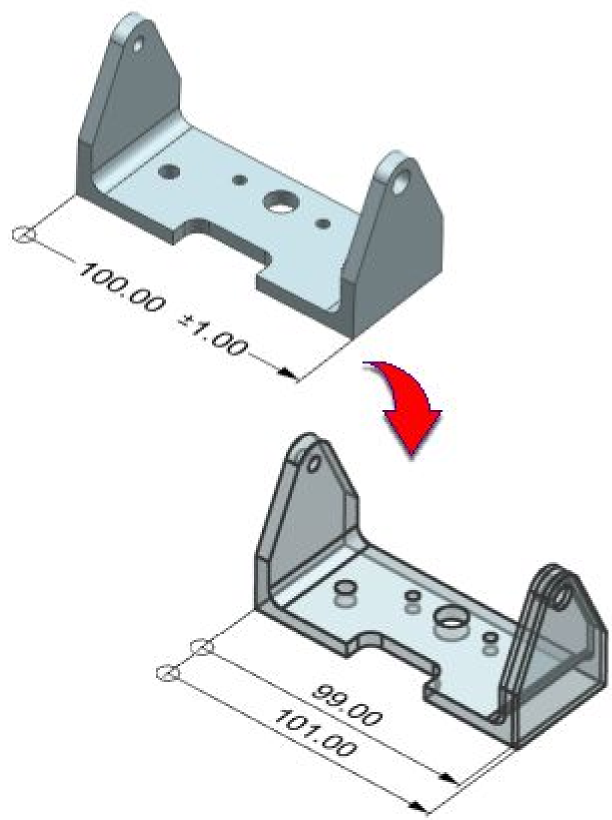
Graphics window	While in the edit mode, click and drag the Arrow Line access handle Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Length**

Linear dimension measurement types

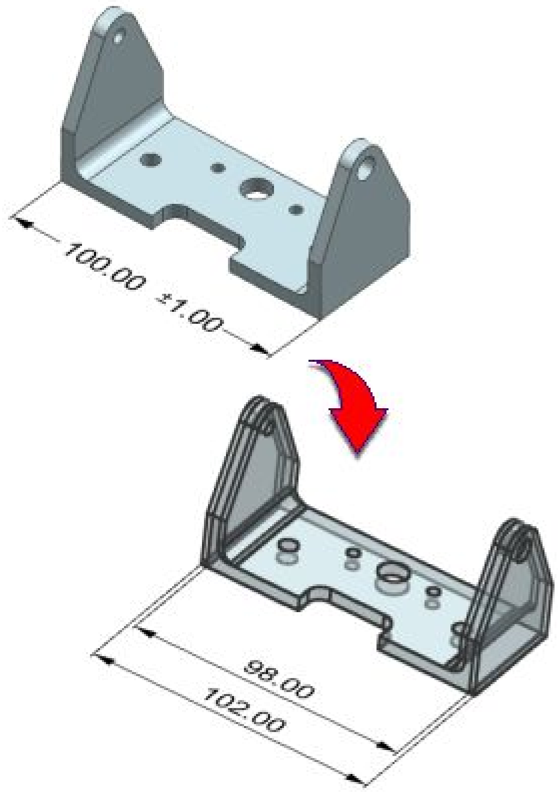
What is it?

A new **Measurement Type** option in the **Linear Dimension** dialog box lets you specify a linear dimension as a general dimension, a directed dimension, or a feature of size dimension. The dimension type determines how the feature is measured and how the tolerance is applied to the measurement. This additional information is saved with the dimension object when it is created.

- A **General** dimension is a simple dimension with no additional, implied information about the measurement.
- A **Directed** dimension specifies the direction in which a feature is measured from an implied datum. All of the tolerance is applied to the second object.



- A **Feature of Size** dimension is a measurement about an implied datum halfway between the two measured features. The tolerance is equally disposed about the two measured features.



When the **Measurement Type** is set to **Directed**, you can select multiple objects for the origin object and the measured object. When it is set to **Feature of Size**, you can select multiple objects for the first object and the second object

Note

> To make the **Measurement Type** option available in the **Linear** dimension dialog box, you must first set the **Enable Directed Dimensions** option in the **Drafting Preferences** dialog box. This option is not available in the **Settings** dialog box, so you cannot set it while creating or editing a linear dimension

Hole callouts, cylindrical dimensions, chain dimensions, and baseline dimensions are not supported as directed or feature of size dimensions.

Why should I use it?

Use the new measurement types to capture additional design information when dimensioning the features on your model. In particular, the information captured for PMI dimensions can be used in downstream applications such as tolerance analysis and inspection applications.

Where do I find it?

Measurement Type option

Application	Drafting, PMI

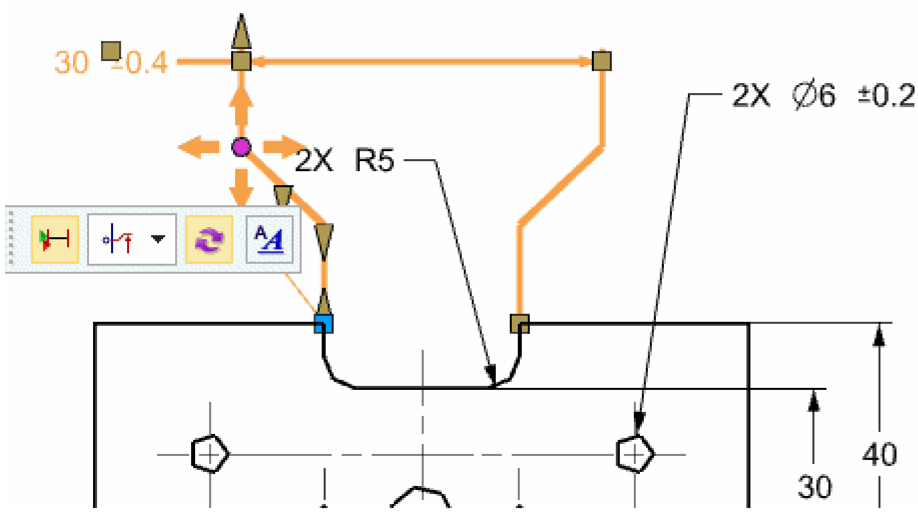
Prerequisite	The **Enable Directed Dimensions** option in the **Drafting Preferences** dialog box must be set.
Command Finder	**Linear**
Location in dialog box	**References** group→**Measurement Type**

Enable Directed Dimensions option

Application	Drafting, PMI
Command Finder	**Drafting Preferences**
Location in dialog box	**Dimensions**→**Workflow** node→**Enable Directed Dimensions**

Jogs on linear dimensions

You can add or remove jogs on extension line while creating or editing linear dimensions. Interactive on-screen handles and controls let you add, place, and size the jog for one extension line or for both extension lines.



Preference and setting options are available to control the initial parameters of the jog.

Where do I find it?

Interactively add or remove jogs

Application	Drafting and PMI
Graphics window	Double-click an existing dimension, or while creating a dimension, click the **Edit** on-screen button to enter the dimension edit mode, and then click the Extension Line access handle to display the on-screen controls.

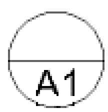
Set the jog parameters

Application	Drafting and PMI
Location in dialog box	In the **Drafting Preferences** dialog box: **Dimension→Jog** node→**Linear Format Side 1** group and **Linear Format Side 2** group In the **Settings** dialog box: **Jog** node→**Linear Format Side 1** group and **Linear Format Side 2** group

Movable datum targets, new terminator types

What is it?

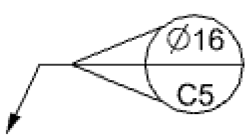
A new **Movable** option lets you add a movable modifier to a datum target symbol.



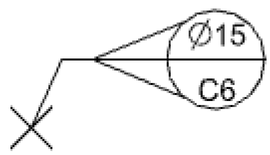
Datum target symbol

Datum target symbol with a movable modifier

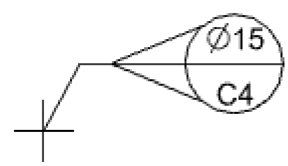
Additionally, you can now specify the following leader termination types for datum targets.



Arrow Only



X



Plus

The display of the movable modifier symbol is compliant with either the ASME or ISO drafting standard. When a standard other than ASME or ISO is in use, the display conforms to the ISO drafting standard. The orientation of the movable modifier may be adjusted when the ISO display is used.

Why should I use it?

Use this type of datum target when the form of the feature is such that an acceptable datum target can be anywhere along a location range. For example, round parts such as cast and forged parts that are fixtured using equalizing datums.

Where do I find it?

Application	Drafting and PMI
Command Finder	**Datum Target**

Location in dialog box	**Target** group→**Movable** option and **Terminator** list

Specifying Item Type for custom symbols

What is it?

In a Teamcenter Integration environment, new options in the **Define Custom Symbol** dialog box and the **Define Symbol from Catalog** dialog box let you specify an **Item Type** in addition to specifying the name, revision, and number for the symbol. You can select the item type from a list of types that are available in the database. This list is typically the same as the list you see when the **Blank** template is selected in the **File New** dialog box.

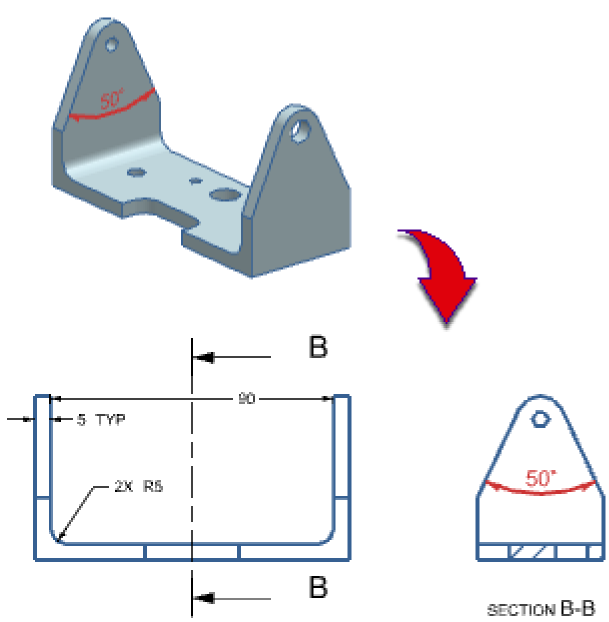
Where do I find it?

Application	Drafting and PMI
Prerequisite	You must be in a Teamcenter Integration environment
Command Finder	**Define Custom Symbol** 
Location in dialog box	**Symbol Name** group→**Item Type**

Inheriting PMI into a Drafting section view

What is it?

You can now use inherit PMI options to add PMI from a 3D model into a drafting section view.



You can add PMI to the following drafting section views:

- Simple
- Stepped
- Half
- Pictorial
- Half Pictorial

All other section views are not supported.

The inherit PMI options set for the parent view are applied when you create the section view.

Note

For existing drafting section views in parts created in earlier versions of NX, you must version up the views before the inherit PMI options are available.

(**Drafting Preferences** dialog box→**General/Setup**→**Workflow**→**Version Up** group)

Why should I use it?

You can inherit PMI in a drafting section view instead of creating a PMI lightweight section view and then adding that view to your drawing. This approach lets you take advantage of reusing existing information and of bi-directionally editing your annotations.

Where do I find it?

Application	Drafting
Graphics window	Right-click the border of an existing Drafting section view→**Settings** →**Common**→**PMI**

Editing property tracking for inherited PMI

What is it?

Use the new **Inherit PMI Tracking** command to interactively change the status of tracked properties of inherited PMI in drawings. Tracked properties are updated in the drawing view when changes are made to the 3D PMI objects.

For example, from the **Inherit PMI Tracking** dialog box, you can enable the **AnnotationLocation** option to keep track of the position changes for all PMI annotation inherited onto the drawing.

If changes to NX result in new PMI properties that are not currently on the property tracking list, you can use the **Update Tracking Properties** option to add these new properties to the list.

Why should I use it?

Use this command to add tracked properties that are saved with the part file, or to modify the tracking status of existing tracked properties of inherited PMI objects on a drawing.

Where do I find it?

Change tracking status with the **Inherit PMI Tracking** command

Application	Drafting

Menu	**Format→PMI→Inherit PMI Tracking**

Add newly trackable properties with the **Update Tracking Properties** option

Application	Drafting
Prerequisite	Open a part that does not include all tracking properties supported by the current version of NX in its set of properties available for tracking.
Menu	**Format→PMI→Inherit PMI Tracking**
Dialog box	**Version Up** group→**Update Tracking Properties**

Information on leader termination points

What is it?

For PMI and Drafting annotation and custom symbols, the XYZ coordinates of leader and extension line terminators that terminate on geometry are now displayed when object information is requested for the selected objects. The coordinate information is displayed as Termination Point data. For example:

```
Layer                1
Type                 PMI Linear Dimension

Extension Line       1
Color                186 (Red)
Font                 Solid
Density              Thin
Termination Point    XC =   14.900000000         X =   14.900000000
                     YC =    0.000000000         Y =    0.000000000
                     ZC =   13.000000000         Z =   13.000000000
```

Reporting of terminator coordinates is subject to the following constraints:

- If the object does not have any leaders, no information is output.
- Information is output for each leader line or extension line of the object that is used and which terminates on geometry or supplemental geometry. No leader termination information is output for leaders that terminate on other PMI or annotation.
- Since the objective is to report the coordinates of the terminator geometry, the point reported does not take into account any gaps between the end of the extension line and the geometry.

Why should I use it?

You may want to use information regarding where the leaders or extension lines connect to geometry in downstream processes.

Where do I find it?

Application	Gateway and Drafting

Prerequisite	The model or drawing must contain PMI or Drafting annotation or custom symbols, or Feature or Sketch dimensions displayed as PMI, that have leaders or extension lines terminating on geometry.
Report window	Create an information report on the above objects using the **Object Information** command and then search for **Termination Point**.

Creating PMI objects from a JT file

What is it?

When you open a JT file in NX, you can create real PMI objects from data in the file. Types of PMI that can be created as real NX PMI from a JT file include all the following:

PMI object type	PMI object subtype
Annotation Note Balloon * PMI Centerline * PMI Centermark * Coordinate Note * Datum Feature Symbol Datum Target	
Dimension	Angular Arc Length * Chamfer * Diametral Folded Radius * Limits & Fits * Linear Radial
Feature Control Frame Line Weld * Model Views Part – Native Part – Managed Assembly – Native Assembly – Managed Surface Finish	

* Added in NX 10

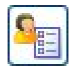
Note

To create real PMI objects from the JT file, the **Create Real PMI Symbols from JT Files** customer default must be set.

Why should I use it?

Creating real NX PMI objects from a JT file simplifies authoring and allows the data to be used in downstream applications that consume PMI.

Where do I find it?

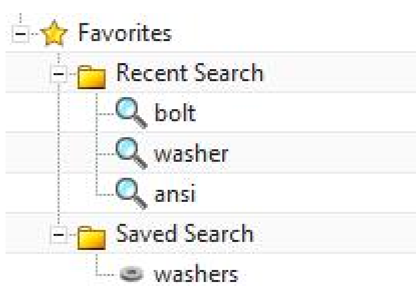
Application	Gateway
Command Finder	**Customer Defaults** 
Location in dialog box	**PMI→Create Real PMI Symbols from JT Files**

Data Reuse

Favorites and Ribbon Bar in the Reuse Library

What is it?

A new **Favorites** node is added to the **Reuse Library** Navigator. It contains two folder nodes where you can store recent searches and saved searches if the Index search is on. The **Saved Search** node is shown only when the Index search is off.



When you perform an Index search in the **Reuse Library**, the search result is displayed, and the search criteria is saved in the **Recent Search** node automatically. The node will remember the previous 30 entries. If you want to save the search criteria permanently, you can save a recent search in the **Saved Search** node, and specify a custom name and image to represent it.

It is not necessary to have the **Index Search Server** in order to add objects to and use objects from the **Saved Search** node

You can add saved searches and objects to either a **Ribbon Tab** or a **Border Bar**. The search result will be shown for the search.

Why should I use it?

You can create reuse library searches based on multiple criteria, and save them for future use.

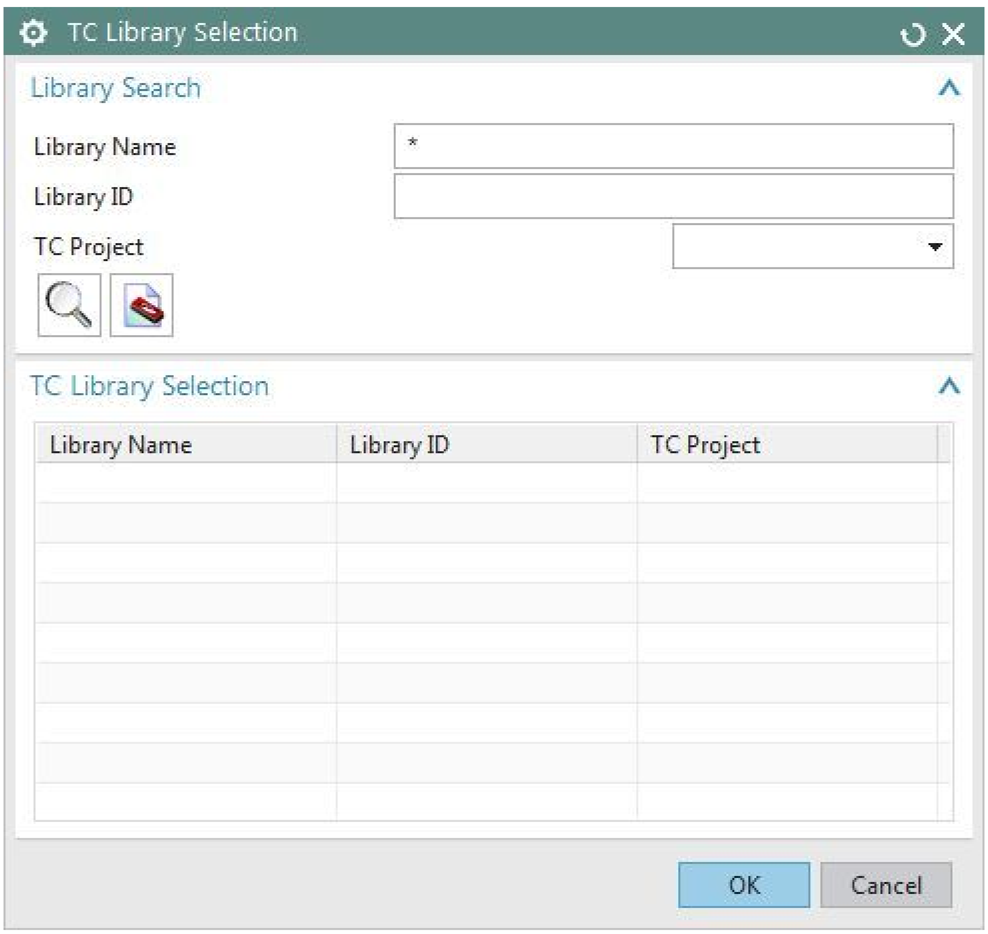
Where do I find it?

Resource bar	**Reuse Library**

Teamcenter library integration

What is it?

You can now use the **Reuse Library Management** command to access the Teamcenter library hierarchy. You can browse and search for library elements and publish them to the Teamcenter library. The Teamcenter library lets you create different library content based on design context.



Why should I use it?

The Teamcenter library integration with NX lets you utilize Teamcenter library features within NX.

Where do I find it?

Prerequisite	You must be in a Teamcenter environment.
Command Finder	**Reuse Library Management**

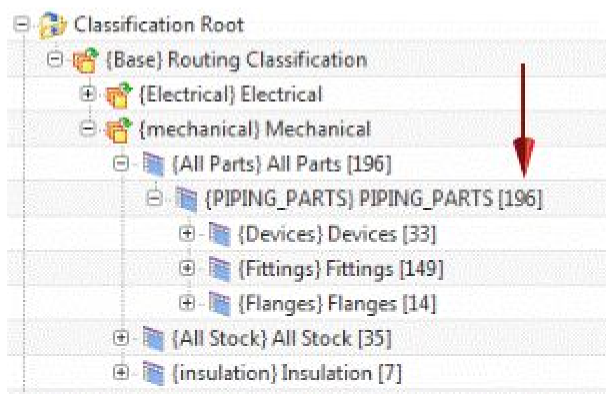
Menu	**Tools→Reuse Library→Reuse Library Management**

Reuse library configuration enhancements

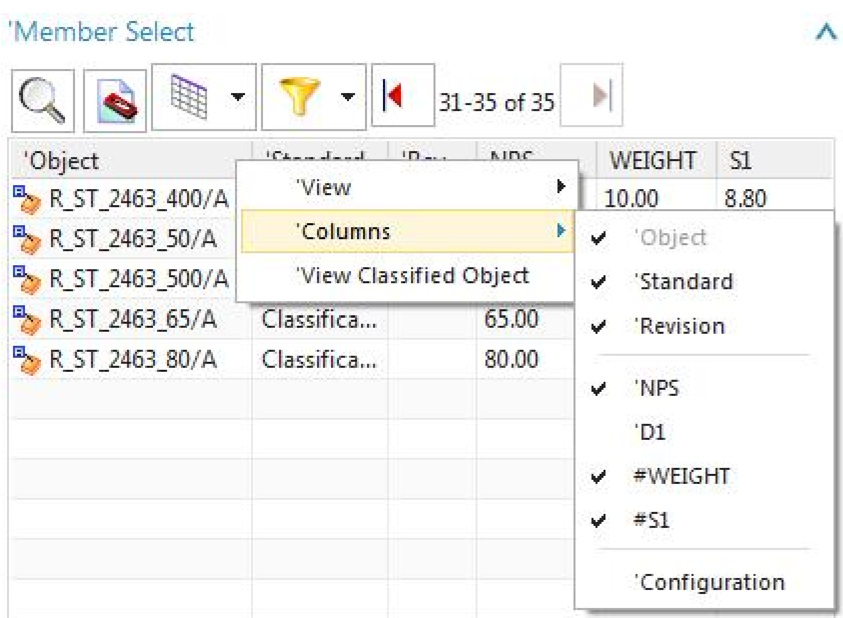
What is it?

There are several enhancements to configuration methods in the **Reuse Library**:

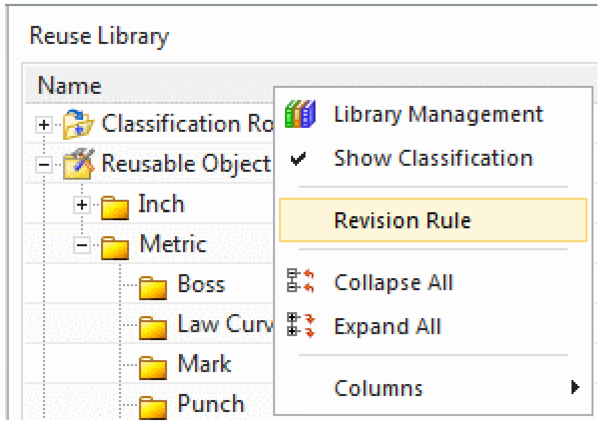
- The number of objects in a classification reuse library is automatically updated when objects are added to a node.



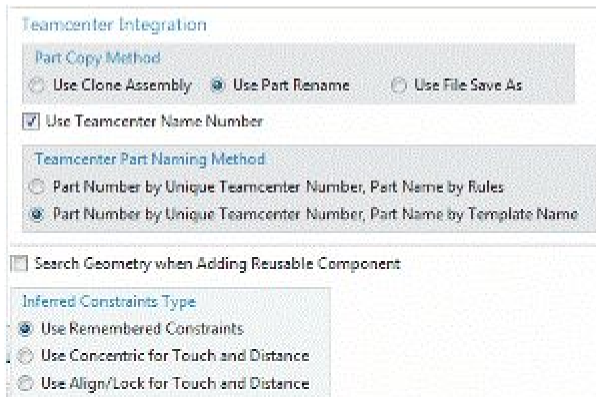
- Classification attributes are displayed in the **Member Select** panel. You can configure the column display for each Classification class when using **Table View** mode.



- You can now set the Teamcenter revision rule in the reuse library navigator. The items in the library are loaded using the Teamcenter revision rule you select.

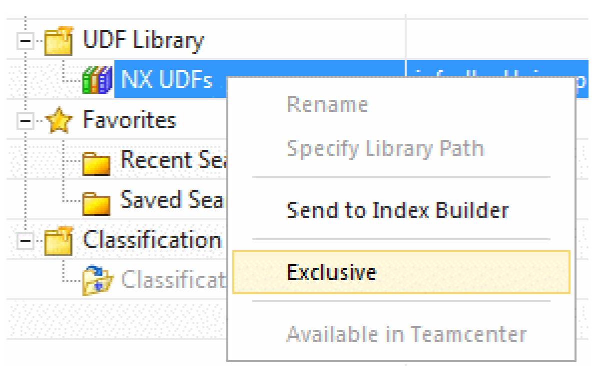


- There are new **Customer Default** options in Teamcenter Integration for reusable component part copy methods, part naming methods, and inferred constraints:

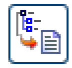


- You can filter the reuse library by application with the **Reuse Library Management** command.

  You can use the **Exclusive** option to assign a library node to be shown only in the specific application that you choose.

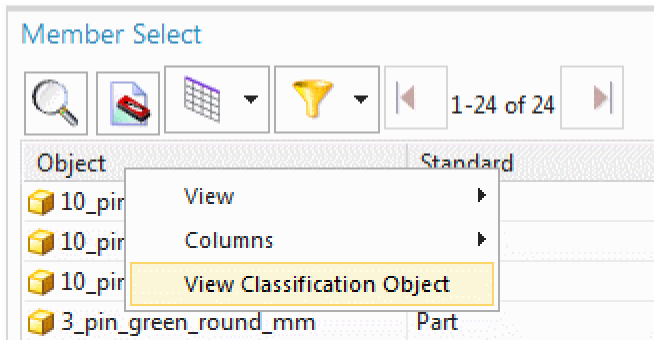


- You can now save your reuse library configuration in a .krx file using the **Export configuration file** command.

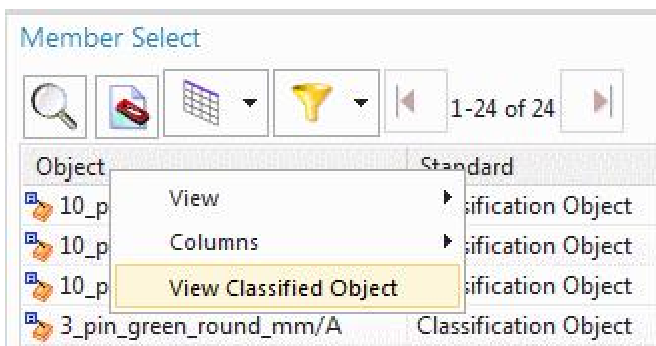


- You can choose to replace or append the current configuration list by selecting **Overwrite the current list when loading configuration** in the **Settings** group.

- You can now set the view mode in the reuse library **Member Select Panel** for classification nodes. If you set the view mode to **View Classification Object**, the reuse library browser does not retrieve the associated Teamcenter objects, and the search performance is improved.

  If you set the view mode to **View Classified Object**, the reuse library determines the classified object type, and displays the preview image.

- Teamcenter multifield key functionality (MFK) is now supported by NX and the reuse library functionality also supports MFK.

Where do I find it?

Prerequisite	You must be in a Teamcenter environment.
Resource bar	**Reuse Library** 

Teamcenter classification reuse library enhancements

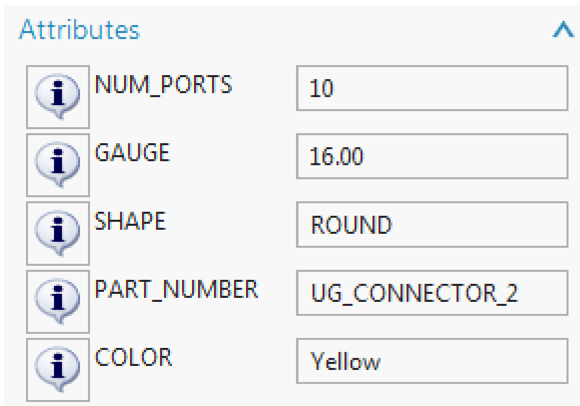
What is it?

You can now search the Teamcenter Classification library for objects based on the following types of attributes:

- Key LOV

- String
- Integer
- Real
- Date

You can specify attribute values when you search for existing objects or when you assign new values to objects using the **Edit Classified Part** command. The following image shows an example of the types of attribute values that you can save or edit for a classification object.



The input values for the attributes are validated according to the attribute type.

Where do I find it?

Prerequisite	You must be in a Teamcenter environment.
Resource bar	**Reuse Library** 

Fastener assembly configuration library

What is it?

You can now use the **Fastener Assembly Configuration Library** reuse library. This new default library is configured with the *FastenerAssemblyConfiguration.krx* file.

You manage fastener assemblies in the **Fastener Assembly Configuration Library** with the **Configure Fastener Assembly** command. You can create, delete, and edit fastener assemblies, and assign unique icons to the assemblies that appear in the reuse library **Member Select** panel.

Why should I use it?

You can search for fastener assemblies in the **Fastener Assembly Configuration Library** and drag them to your part.

Where do I find it?

Command Finder	**Configure Fastener Assembly**

Resource bar	**Reuse Library**

Index search from Reuse Navigator

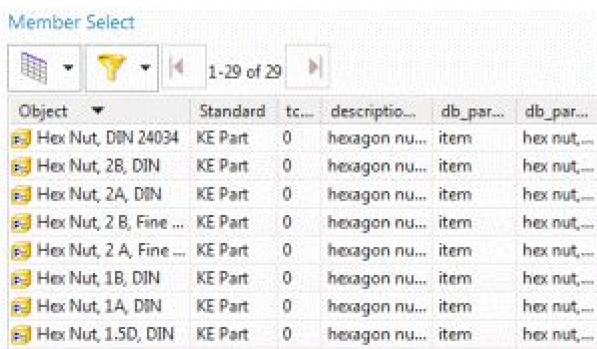
What is it?

If a reuse library contains index data, you can use the **Build by Index Data** command to create the reuse library tree from index data.

A new SOLR installation tool named **Siemens PLM Documentation Server** is used to set up the index search.

Why should I use it?

You can increase the performance of browsing if you create a reuse navigator index for a library. Values for part attributes or part expressions can be shown in the reuse library **Member Select** panel if the values are indexed.



The new installation tool can be used to set up index search server.

Where do I find it?

Command Finder	**Build by Index Data**

Preferred Data

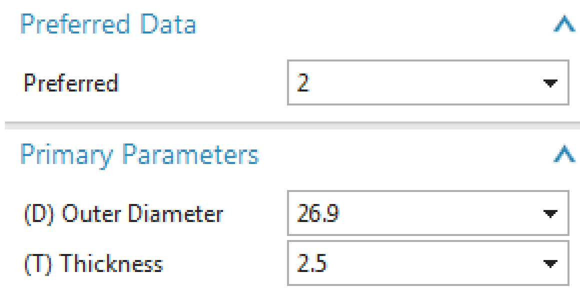
What is it?

You can include a column named **Preferred** in a spreadsheet for a reusable component. You can assign values in the **Preferred** column to specify which reusable components have first preference.

PARAMETERS			
BODY_THICKNESS	BODY_OUTSIDE_DIAMETER	LENGTH	PREFERRED
2	21.3	50	1
2.5	21.3		1
3	21.3		2
2	26.9		2
2.5	26.9		2
3	26.9		3

If the **Preferred** column exists in the spreadsheet, a **Preferred Data** list is displayed in the **Reuse Paste** or **Add Reusable Component** dialog box.

When you specify a **Preferred** reusable component, only the data series that matches the preferred value will be shown in the dialog for you to choose.



Why should I use it?

You can control the number of reusable objects and reusable components to choose from if you filter them to show only those with preferred attributes.

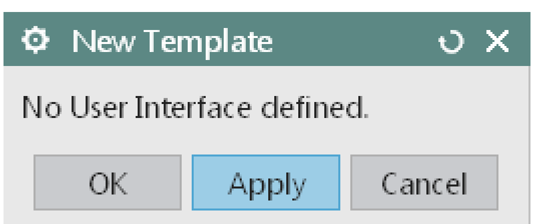
Where do I find it?

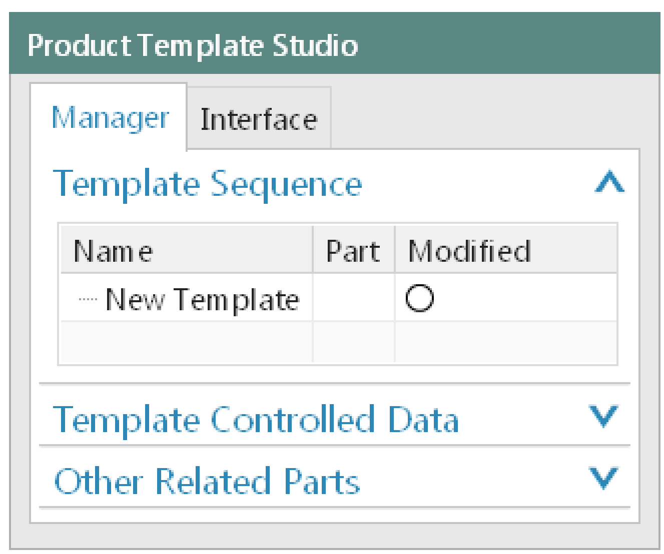
Location in dialog box	**Reuse Paste**, **Add Reusable Component**

Product Template Studio user interface enhancement

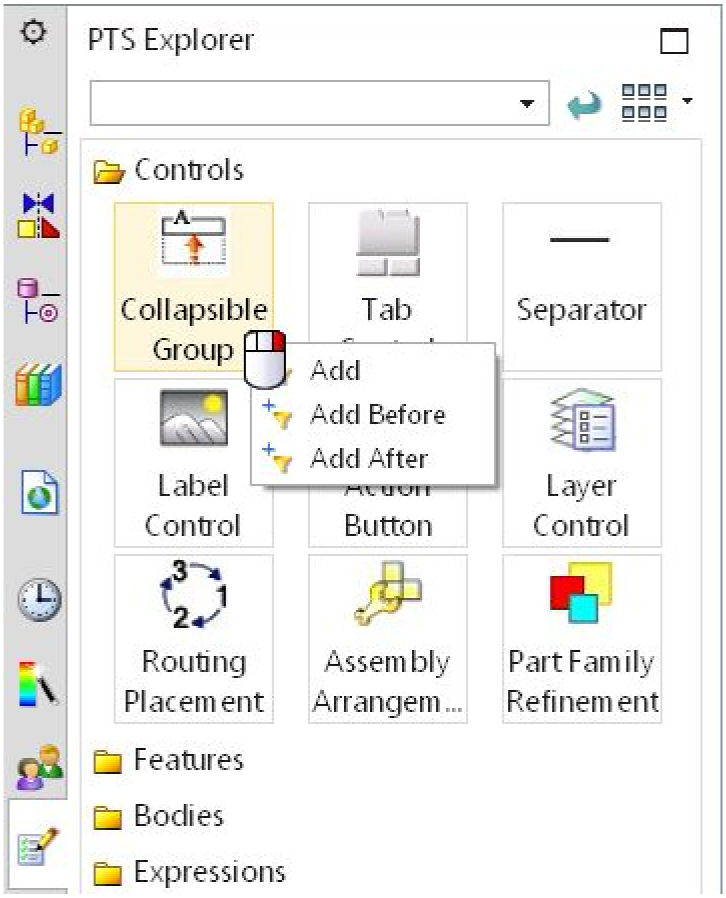
What is it?

You can now author and edit Product Template Studio templates in NX using the **PTS Author**  application. When you launch the application, the following dialog boxes open and the **PTS Explorer**  displays in the **Resource Bar**.

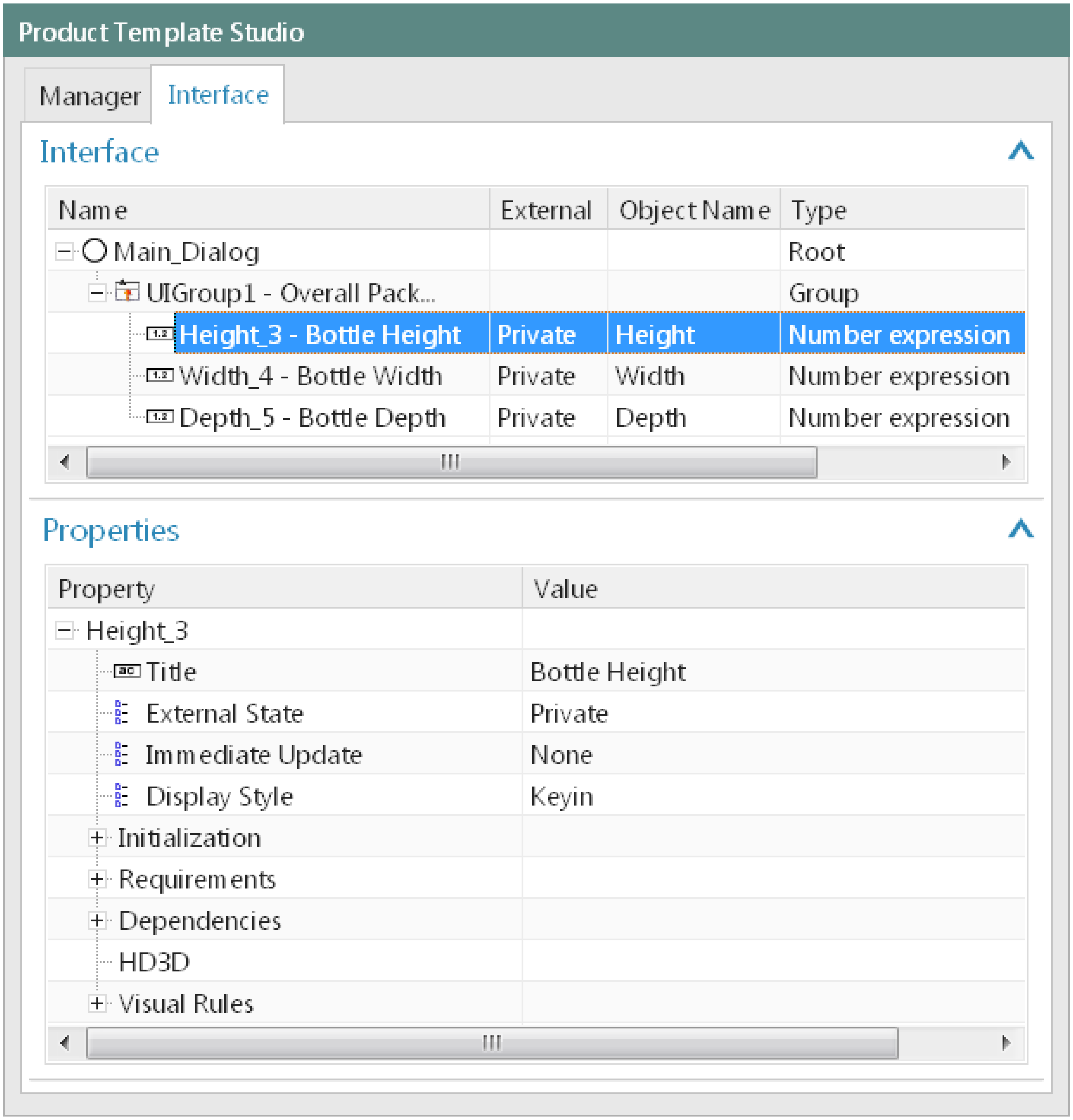




In the **PTS Explorer** you can right-click objects from the folders to add them to the template.



In the **Product Template Studio** dialog box, on the **Interface** tab you can modify the **Properties** of the objects that you add to the template.



Where do I find it?

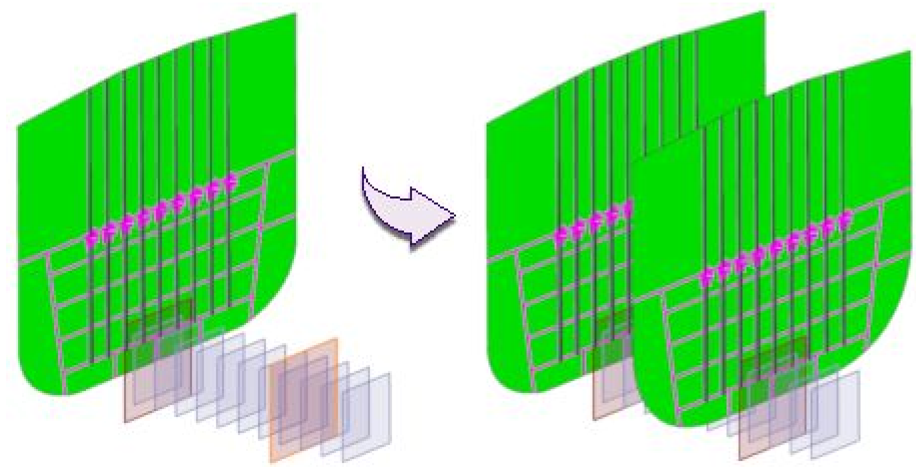
Application	PTS Author

Ship Structure

Copy Basic Design Objects

What is it?

Use the **Copy Basic Design Objects** command to copy basic design objects such as plate systems, pillar systems, stiffener systems, standard part brackets, cutouts, and seams.



You can:

- Copy a plate system or a pillar system to another location.
- Copy the following types of features attached to a plate system to another plate system:
  - Stiffener systems
  - Edge reinforcement systems
  - Standard part brackets
  - Cutouts
  - Extrude features
  - Seams

Why should I use it?

This command saves time by letting you quickly copy repetitive structures to different locations in a basic design.

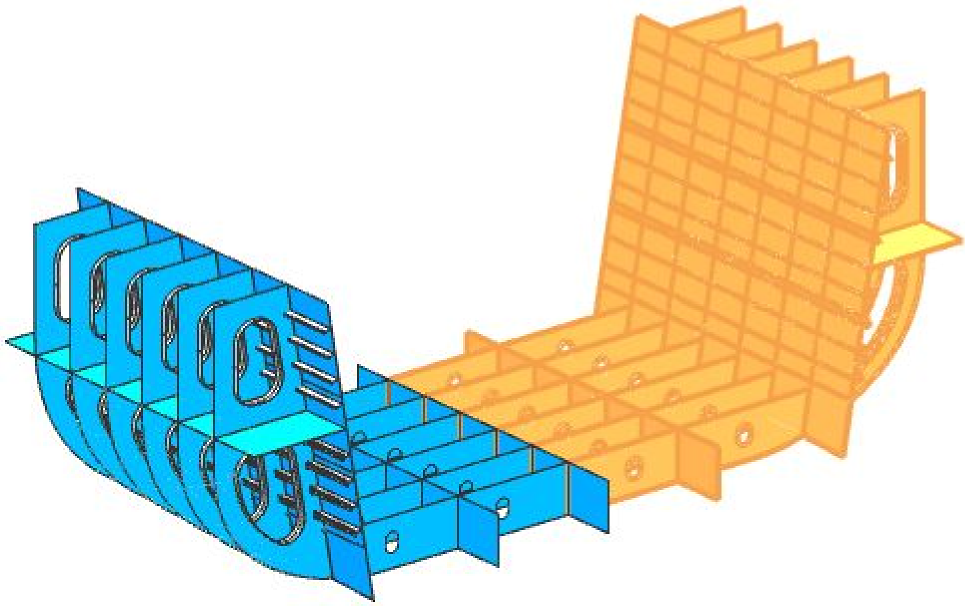
Where do I find it?

Application	Ship Structure Basic Design
Command Finder	**Copy Basic Design Objects**

Mirror Ship Structure

What is it?

Use the **Mirror Ship Structure** command to mirror detailed design elements or components through a plane. The resulting features in the mirrored design elements or components are not associated to the source features and you can edit them independently.



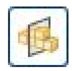
You can:

- Mirror the following types of features:
  - o Plate
  - o Stiffener
  - o Edge Reinforcement
  - o Pillar
  - o Standard part brackets and collar plates
  - o Cutout
  - o Corner Cut
  - o Edge Cut
  - o Along Guide Cut
  - o Plate Chamfer
  - o Extrude
  - o End Cut
  - o Profile Cutout
- Specify a mirror plane or use the default longitudinal vertical center plane of the ship.
- Generate a report in an XML file that lists which parts were successfully mirrored, which parts failed, the duration of the operation, and other detailed information.

Why should I use it?

You can save a lot of time by quickly copying repetitive, symmetrical structures in a ship. This command is more flexible than the existing **Mirror Feature** and **Mirror Assembly** commands because you edit and update the mirrored geometry as if it was created separately.

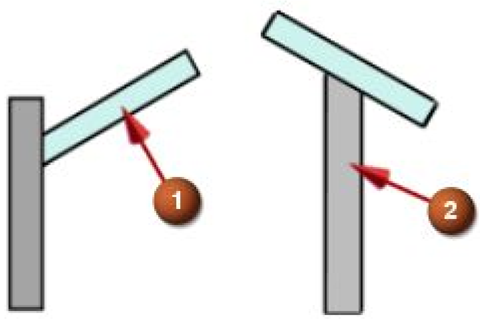
Where do I find it?

Application	Ship Structure Detail Design
Prerequisite	A ship container part must be identified in the displayed part. If a ship container part is not defined, the **Ship Container Part** dialog box is displayed automatically when you use this command.
Command Finder	**Mirror Ship Structure** 

Edge reinforcement and stiffener enhancements

What is it?

The **Stiffener/Edge Reinforcement** and **Edge Reinforcement System** commands are enhanced so that you can extend or trim the edge reinforcement (1) or the reinforced plate (2) to produce a neat trim. This supports the automatic creation of weld joints.



The **Stiffener/Edge Reinforcement** command also includes additional enhancements so that you can:

- Create closed loop edge reinforcements. You can then use the **Split Ship Structure** command to define the ends of the edge reinforcements so they can be manufactured.

- Use the same **On Edge** and **On Face** placement types that are available with the **Edge Reinforcement System** command. This replaces the existing offset value option.

Why should I use it?

The enhancements let you create structures that support downstream operations such as welding and manufacturing and they provide consistency between commands in Ship Structure Basic Design and Ship Structure Detail Design applications.

Where do I find it?

Application	Ship Structure Basic Design
Command Finder	**Edge Reinforcement System**

Application	Ship Structure Detail Design
Command Finder	**Stiffener/Edge Reinforcement**

Paint Parameters enhancements

What is it?

The **Paint Parameters** command is enhanced so that you can copy parameters from standard parts and additional types of features.

You can select the following types of features in the Ship Structure Basic Design application:

- **Profile Cutout**
- **Cutout**
- **Standard Part** (brackets and collar plates)

You can select the following type of feature in the Ship Structure Manufacturing application:

- **Excess Material**

You can select additional types of objects in the Ship Structure Detail Design application:

- **Profile Cutout** features
- **Cutout** features
- **Plate Chamfer** features
- **Along Guide Cut** features
- **Standard Part** (bracket and collar plate components or design elements)

Why should I use it?

You can save time by applying the same parameters to multiple features at once.

Where do I find it?

Application	Ship Structure Basic Design, Ship Structure Detail Design, and Ship Structure Manufacturing
Command Finder	**Paint Parameters**

Prevent circular dependencies in a basic design

What is it?

The commands in the Ship Structure Basic Design application are enhanced to prevent circular dependencies. In previous versions, when you defined boundaries and seams to establish relationships between plate systems, a circular dependency could produce the following error message:

```
There is an object which depends on itself.
```

Why should I use it?

You no longer have to create duplicate plate systems to work around circular dependencies.

Where do I find it?

Application	Ship Structure Basic Design
Command Finder	The enhancements do not change any command dialog boxes. They affect the processing that occurs in the background when you use a command that creates or updates plate systems.

Ship Structure performance enhancements

What is it?

The following enhancements are implemented to improve performance when you use the Ship Structure Basic Design and Ship Structure Detail Design applications:

- Unnecessary interpart updates are eliminated. For example, when you add a corner cut to a plate, the stiffeners that are placed on the plate no longer update.

- **Standard Part** enhancements:
  - o In the Ship Structure Basic Design application, the **Standard Part** dialog box includes a **Type** option to let you adopt an existing curve. This improves performance because the necessary parameters and attributes are assigned to simple curves and the standard part templates do not have to be loaded. The detailed standard part is generated when you transition the basic design to the detailed design.
  - o The **Standard Part** reference set is removed from the standard part template. Using the existing **Model** reference set improves performance when you design in context, generate a drawing, and search by volume.
  - o The standard part templates and parameters are simplified.
  - o The option to create standard parts across plates with multiple thicknesses is disabled by default.
- Enhancements specific to the Ship Structure Basic Design application:
  - o When you create a plate system, a solid body is no longer generated since it is not referenced by other structures.
  - o A **Mounting Method** option is added to the **Stiffener System** dialog box and the setting is saved so that it can be used when you transition parts to the detailed design.
  - o The time it takes to display the **Stiffener System** dialog box is reduced.
- **Transition Parts** enhancements:
  - o The automatic update of the detailed design is eliminated when you edit a feature in the basic design. The links from the basic design to the detail design are isolated so the update only occurs when you transition the parts.
  - o Updates are minimized when you transition parts to the detailed design using the **Update Existing** option.
- **Stiffener/Edge Reinforcement** enhancements:
  - o Updates that occur when you access the **Stiffener/Edge Reinforcement** dialog box are eliminated.
  - o The time it takes to create hull stiffeners using the **Normal to Surface** orientation method is reduced.
- Redundant updates are reduced when you access spreadsheet-driven steel features.
- Unnecessary updates are eliminated when you fully load parts.
- Performance is improved in the Teamcenter Integration for NX interface.

Why should I use it?

The enhancements reduce the time it takes to create and update ship structures.

Where do I find it?

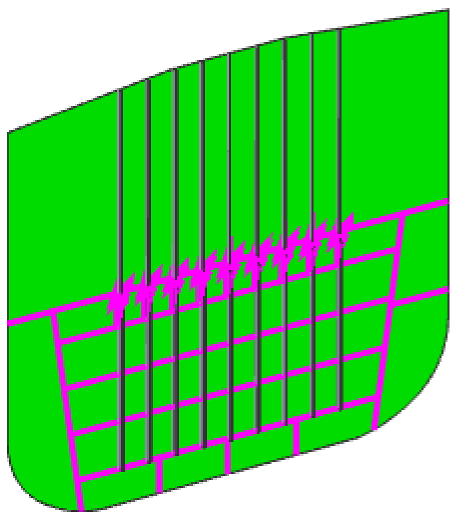
Application	Ship Structure Basic Design and Ship Structure Detail Design
Command Finder	The enhancements affect various commands that you can use to access, create, edit, or update ship structure features and parts.

Weight and Center of Gravity

What is it?

Use the **Weight and Center of Gravity** command to calculate weight and center of gravity information for each subsystem in selected ship structure systems or volumes in a basic design.

The results are reported in a spreadsheet that you can easily use to perform other calculations. In this example, a transverse bulkhead part is selected and portions of the resulting spreadsheet are shown.



		Weight and Center of Gravity Report		
Selection Type:	Frame			
Part	System	SubSystem	Weight(Kilograms)	Center of Gravity(mm,mm,mm)
TBH00007/A			79521.702	(87989.916,1.014,11561.951)
	StiffenerSystem(21)		707.534	(87799.382,19.478,11475.195)
		Subsystem_1	411.122	(87776.542,19.164,6825.000)
		Subsystem_2	296.412	(87831.061,19.915,17925.000)
	StiffenerSystem(33)		703.573	(87799.203,5980.524,11405.981)
		Subsystem_1	411.122	(87776.541,5980.836,6825.000)
		Subsystem_2	292.451	(87831.060,5980.085,17845.833)
	TransverseBulkhead(3)		73178.038	(88006.443,0.906,11573.512)
		Subsystem_1	34684.448	(88006.000,0.000,17750.111)
		Subsystem_2	4179.618	(88006.000,13742.666,6322.555)
		Subsystem_3	1082.421	(88006.000,7884.968,1017.651)

Why should I use it?

This command lets you quickly determine the weight and center of gravity of individual systems and subsystems without having to manually modify or extract geometry.

Where do I find it?

Application	Ship Structure Basic Design
Command Finder	**Weight and Center of Gravity** 

Material Estimation

What is it?

Use the **Material Estimation** command to estimate the material requirements for selected ship structure systems or volumes in a basic design.

The results are reported in a spreadsheet and include the surface area of required plate stock and length of the required profile stock. In this example, a transverse bulkhead frame part is selected.

		Material Estimation Report			
System Type:	Material	Grade	Thickness(mm)	Surface Area(Square Meter)	
Plate					
	Steel	A H24	12	614.595	
	Steel	A H24	18	60.333	
	Steel	A H24	14	63.195	
	Section Type	Size	Material	Grade	Length(Meter)
Stiffener					
	AG	AG-350X100X11/17	Steel	A H24	86.85
	AG	AG-250X90X8/14	Steel	A H24	105.683

Why should I use it?

This command lets you quickly determine the amount of material required for systems and use the results to perform additional calculations.

Where do I find it?

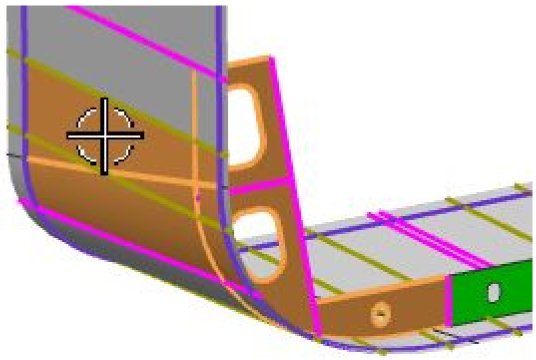
Application	Ship Structure Basic Design
Command Finder	**Material Estimation** 

Unfolded Minimum Rectangle

What is it?

Use the **Unfolded Minimum Rectangle** command to calculate the minimum dimensions of a rectangle that will fit each selected plate subsystem or standard part in the basic design.

The results are reported in a spreadsheet. In this example, three subsystems in a transverse bulkhead and two subsystems in a hull are selected.



	Unfolded Minimum Rectangle Report		
Part	System	SubSystem	Rectangle(Meter X Meter)
TBH000016/A			
	TransverseBulkhead(6)		
		Subsystem_6	5.982X2.006
		Subsystem_7	4.988X4.216
		Subsystem_8	7.213X4.166
Hull000001/A			
	Hull(29)		
		Subsystem_90	14.054X4.41
		Subsystem_91	14.044X3.046

Why should I use it?

This command helps you determine if the unfolded plates will fit on available plate stock and evaluate the locations of straking seams.

Where do I find it?

Application	Ship Structure Basic Design
Command Finder	**Unfolded Minimum Rectangle** 

Edit Context Attributes

What is it?

Use the **Edit Context Attributes** command to modify the attribute assigned to systems, subsystems, and standard parts in the basic design that defines their purpose in the ship. This attribute is inherited by the detailed parts when the basic design is transitioned to the detailed design.

For example, if a stiffener system contributes to the longitudinal strength of the ship, you can change the value of its context attribute to **Longitudinal Strength**. After the stiffener system is transitioned to the detailed design, the resulting stiffeners can be identified so that you can apply appropriate names, welds, and inspection requirements.

The title of the context attribute is **SAW_SUB_NAME**. You can define the allowed values in an XML file located at *%NXSHIP_DIR%\data\NX_ShipAttribute.xml*. When you edit the context attribute, you can select one of the values from a list in the dialog box.

For example, the default list for a stiffener system includes:

- **Default Stiffener System**
- **Longitudinal**
- **Transverse**
- **Vertical**
- **Foundation**
- **Header**

Why should I use it?

Use context attributes to apply ship characteristics to systems, subsystems, and standard parts early in the design process so that they can be used in downstream rules to name detailed parts, add end cuts, add profile cutouts, create collar plates, or create welds.

Where do I find it?

Application	Ship Structure Basic Design
Command Finder	**Edit Context Attributes**

Edit Stock enhancements

What is it?

The **Edit Stock** command is enhanced so that you can change the **Tightness** parameter for non-split plate systems and subsystems.

Why should I use it?

You can apply a unique tightness parameter to selected subsystems within a basic design without having to change the work part and edit features individually.

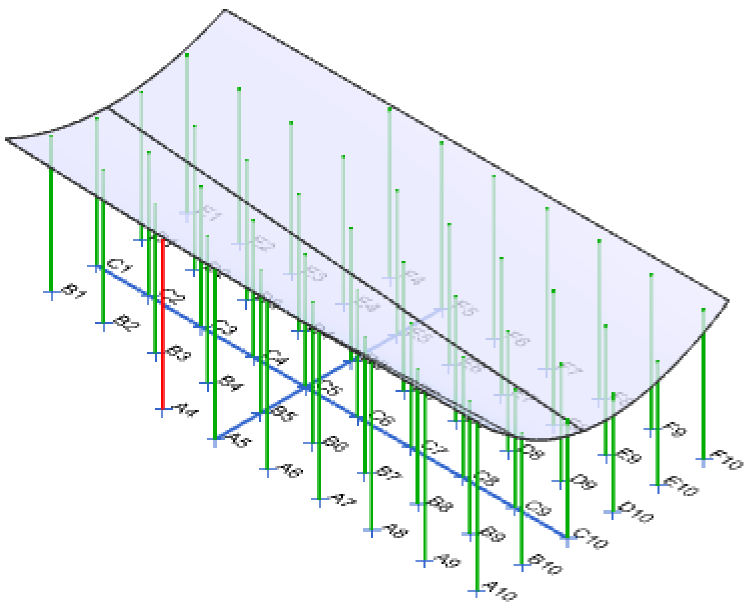
Where do I find it?

Application	Ship Structure Basic Design
Command Finder	**Edit Stock** 
Location in dialog box	**Plate System Stock** group→**Tightness** **Profile System Stock** group→**Tightness**

Pinjig

What is it?

Use the **Pinjig** command to position plates relative to the shop floor and generate pin jig setup data for manufacturing.



You can:

- Define a grid of vertical adjustment pins including the number and spacing in the X and Y directions, minimum and maximum heights, and weld clearance. You can define default pin jigs and parameters in a spreadsheet.

- Select a set of connected plates to place on the pin jig. You also have the option to select additional ship structures attached to plates.

- Define the position and orientation of the plates on the pin jig. You have the option to select an edge from the selected plates to specify the default X direction of the pin jig.

- Display two pin jig reference lines that are shown on the pin grid and are marked on the selected plates. These lines are used to position the plates on the pins.

- Display a table containing the list of pins that touch the plates and their heights.

						A4	4238.65	A5	4207.36	A6	4187.14	A7	4181.13	A8	4181.13	A9	4181.13	A10	4181.13
B1	3898.03	B2	3816.25	B3	3736.54	B4	3659.22	B5	3584.79	B6	3513.85	B7	3447.33	B8	3386.38	B9	3332.16	B10	3285.81
C1	3679.70	C2	3586.14	C3	3491.98	C4	3397.53	C5	3303.20	C6	3209.46	C7	3117.02	C8	3027.05	C9	3000.00	C10	3000.00
D1	3701.60	D2	3608.91	D3	3515.10	D4	3419.88	D5	3323.17	D6	3224.97	D7	3125.47	D8	3025.58	D9	3000.00	D10	3000.00
E1	3957.79	E2	3869.21	E3	3782.27	E4	3696.26	E5	3610.73	E6	3525.30	E7	3438.66	E8	3349.55	E9	3256.21	E10	3155.77
						F4	4220.67	F5	4151.56	F6	4086.92	F7	4023.62	F8	3956.66	F9	3879.51	F10	3783.65
27	000101																		
27	000100																		

- Create Product and Manufacturing Information (PMI) notes to show the distances from the corners of the plates to the closest pins.

Why should I use it?

A pin jig is a common work center used in a shipyard to support plates for welding. This command lets you easily position the plates and define the setup data for the grid of vertical adjustment pins.

Where do I find it?

Application	Ship Structure Manufacturing
Prerequisite	A ship container part must be identified in the displayed part. If a ship container part is not defined, the **Ship Container Part** dialog box is displayed automatically when you use this command. Use the **Cutting Side Face** command to specify the cutting sides of the plates.
Command Finder	**Pinjig** 

Plate Template enhancements

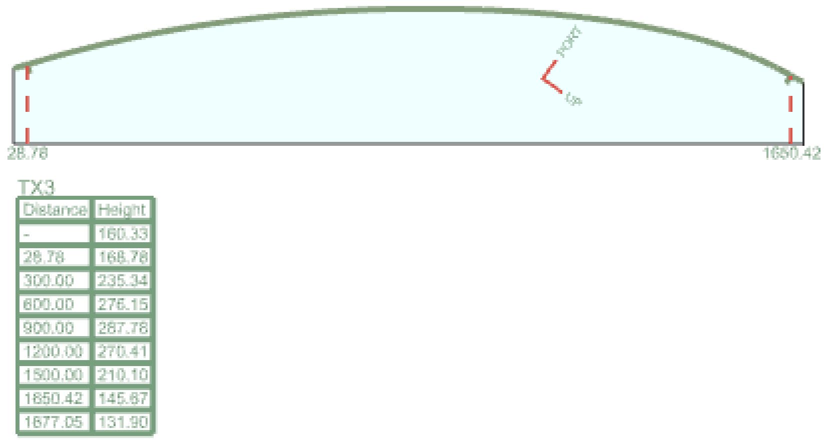
What is it?

The **Template** command is renamed to **Plate Template** and includes several enhancements.

You can:

- Create alignment marks, ship direction marks, base lines, sight lines, and template location marks on the plate.

- Define the maximum height value for the templates. You can also create a table of height values for the resulting templates.



Distance	Height
-	160.33
28.78	168.78
300.00	235.34
600.00	276.15
900.00	287.78
1200.00	270.41
1500.00	210.10
1650.42	145.67
1677.05	131.90

- Create parts, XML output files, drawing views, and offsets for each template.

The dialog box includes the following changes:

- The **Type** group is removed. A manual type is assumed if you define a base coordinate system.
- The **Existing** group is renamed to **Base**. You can define a base coordinate system to align the X and Y templates in the set.
- The **Location** group is renamed to **Locations** and the names of the options are more descriptive. You can select existing planes or define evenly spaced planes for the X and Y template locations.
- The **End Offsets** group is renamed to **Extensions** and the names of the options are more descriptive. You can extend the start and end of the X and Y templates beyond the edges of the plate.

Why should I use it?

These enhancements provide new options and simplify existing options to help you identify and mark sets of templates that match the contour of a formed plate.

Where do I find it?

Application	Ship Structure Manufacturing
Prerequisite	A ship container part must be identified in the displayed part. If a ship container part is not defined, the **Ship Container Part** dialog box is displayed automatically when you use this command.
Command Finder	**Plate Template**

Specifying default attributes and parameters for templates and marking lines.

Command Finder	**Customer Defaults**
Location in dialog box	**Ship Design→Manufacturing→Plate Template** tab

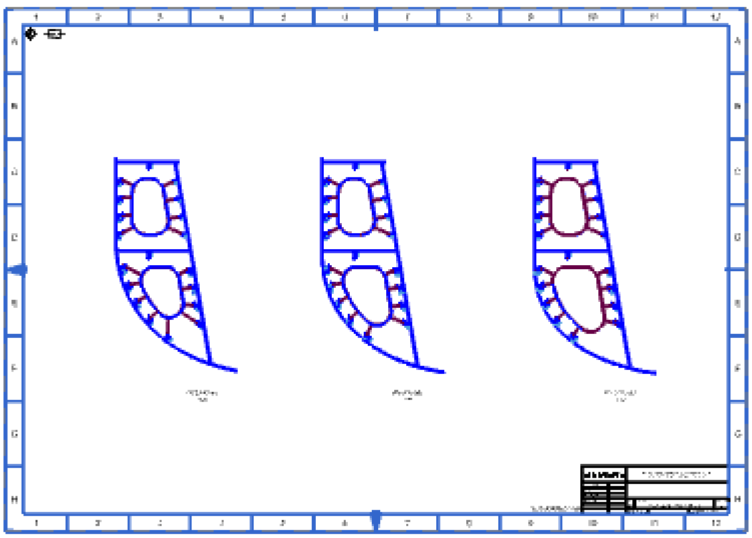
Section Drawing

What is it?

Use the **Section Drawing** command to create deck and bulkhead drawings that typically include the following objects:

- A deck or bulkhead composed of multiple plates.
- Stiffeners that are attached to or pass through the deck or bulkhead.
- Edge reinforcements around openings or other free edges.
- Plates that are bounded by or penetrate the deck or bulkhead.

To reduce the complexity of the drawings, the objects are displayed as symbolic representations that conform to shipbuilding conventions.



You can:

- Select existing frames or specify new planes to define section views.

- Create a single non-master drawing file for all section views or a separate non-master drawing file for each section view. If the displayed part is a non-master drawing file and **Sections per Drawing** is set to **Multiple**, a new sheet is added to the existing non-master drawing file.

- Select **Use Booklets** in the **Ship Drafting** customer defaults to create non-master drawing files as booklets containing worksets.

- Specify a distance value for the **Included Part Proximity** option under **Ship Section** in the customer defaults to automatically determine which objects are hidden in the section view. When you create the drawings, you can also manually select additional objects within the proximity range to hide or select objects outside of the proximity range to include in the section views.

- Specify a different color, font, and width for each type of ship structure.

  - o The initial default values are defined in an XML file located at *%NXSHIP_DIR%\data\NX_ShipAttribute.xml*. This file is also used to assign the classification type and subtype to ship structures using the SAW_NAME and SAW_SUB_NAME attributes.

- You can change the default color, font, and width for each type of ship structure from the **Ship Drafting→View→Ship Structure Lines** node in the **Drafting Preferences** dialog box.

- You can edit the color, font, and width of ship structures in existing section views from the **View→Ship Structure Lines** node in the **Settings** dialog box.

- You can use custom line fonts specified in the font definition file at *%NXSHIP_DIR%\shipdrafting\symbol\ShipLineFontSettings.fdf*. By default, this file includes a new **RAILWAY** font that is available in the Ship Structure applications.

Why should I use it?

Create drawings for multiple decks and bulkheads that are simplified and easier to annotate.

Where do I find it?

Application	Ship Structure Detail Design
Command Finder	**Section Drawing**

Settings to use booklets and define the part proximity

Command Finder	**Customer Defaults**
Location in dialog box	**Ship Drafting→General** **Ship Drafting→Ship Section**

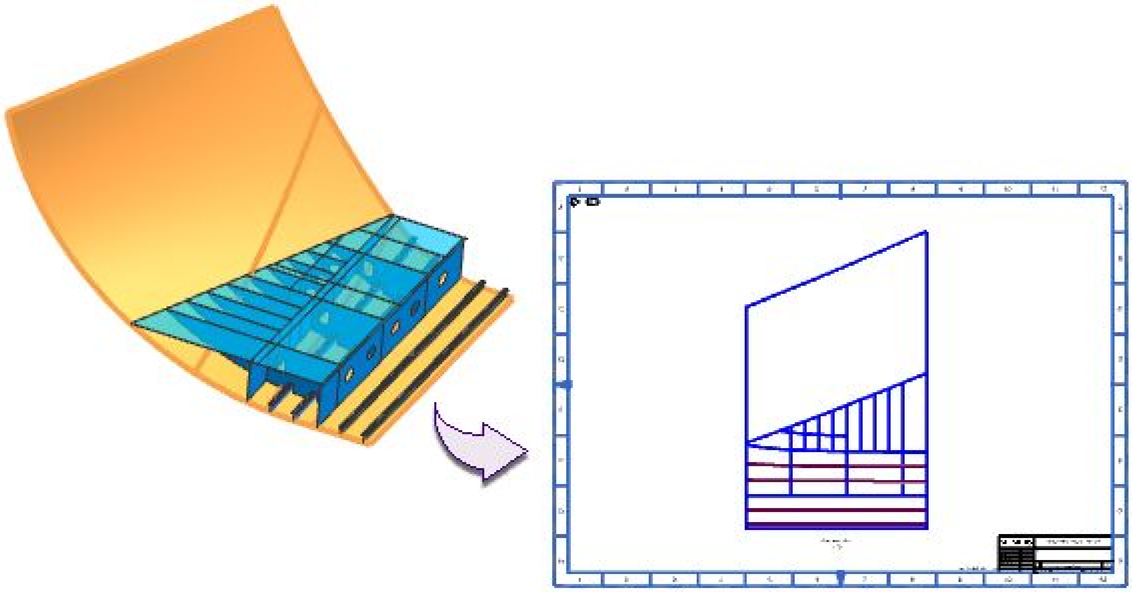
Color, font, and width preferences

Application	Drafting
Command Finder	**Drafting Preferences**
Location in dialog box	**Ship Drafting→View→Ship Structure Lines** node

Expansion Drawing

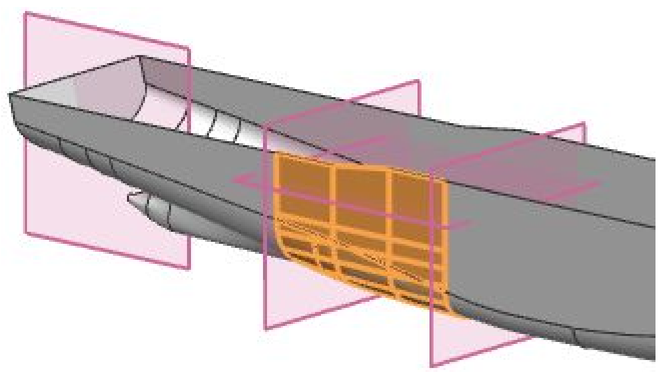
What is it?

Use the **Expansion Drawing** command to create drawings representing non-planar objects, such as hull plates and stiffener landing curves, which are mapped to a view plane. These drawings can also include other planar objects that are not parallel to the view plane.



You can:

- Select multiple connected plates.
- Specify a base plane and optional upper, fore, and aft limit planes.



- Create a new non-master drawing file and drawing sheet for the expansion view. If the displayed part is a non-master drawing file, a new sheet is added to the existing non-master drawing file.
- Specify a different color, font, and width for each type of ship structure.
  - o The initial default values are defined in an XML file located at *%NXSHIP_DIR%\data\NX_ShipAttribute.xml*. This file is also used to assign the classification type and subtype to ship structures using the SAW_NAME and SAW_SUB_NAME attributes.
  - o You can change the default color, font, and width for each type of ship structure from the **Ship Drafting→View→Ship Structure Lines** node in the **Drafting Preferences** dialog box.
  - o You can edit the color, font, and width of ship structures in existing views from the **View→Ship Structure Lines** node in the **Settings** dialog box.

- o You can use custom line fonts specified in the font definition file at *%NXSHIP_DIR%\shipdrafting\symbol\ShipLineFontSettings.fdf*. By default, this file includes a **RAILWAY** font that is only available in the Ship Structure applications.

This command is similar to the existing **Shell Expansion** command but it lets you create a drawing using a drawing template.

Why should I use it?

Create drawings for non-planar ship structures that are simplified and easy to annotate.

Where do I find it?

Application	Ship Structure Detail Design
Prerequisite	A ship container part must be identified in the displayed part. If a ship container part is not defined, the **Ship Container Part** dialog box is displayed automatically when you use this command.
Command Finder	**Expansion Drawing** 

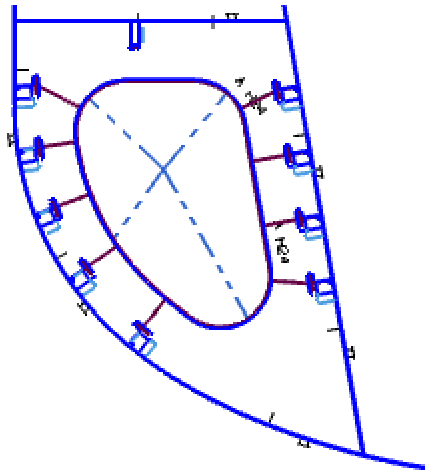
Color, font, and width preferences

Application	Drafting
Command Finder	**Drafting Preferences** 
Location in dialog box	**Ship Drafting→View→Ship Structure Lines** node

Annotate Ship Drawing

What is it?

Use the **Annotate Ship Drawing** command to add symbols and annotation to ship drawing views.



FR68_FW_0@1
1:50

PLATES: = 12mm A H24
STIFFENERS: = FL-100X10 A H24

The symbols are defined in the **Reuse Library** and configured using a spreadsheet specified in the customer defaults. NX provides a set of example symbols and a configuration file located at *%NXSHIP_DIR%\shipdrafting\symbol\drawing_annotation_symbols.xlsx*. The example symbols include:

- Plate and stiffener identifiers
- Scantling text
- Boundary seam symbols
- Plate continuity symbols
- Stiffener identifiers
- Stiffener section symbols
- Opening fill lines
- Thickness direction symbols
- End cut symbols

Why should I use it?

Automatically create symbols and other annotation that are unique to ship structures.

Where do I find it?

Application	Drafting
Prerequisite	Create drawings of ship structures using the **Section Drawing** and **Expansion Drawing** commands.
Command Finder	**Annotate Ship Drawing**

Specifying the location of the drawing annotation symbol configuration file.

Command Finder	**Customer Defaults**
Location in dialog box	**Ship Drafting→Annotation→Symbol** tab

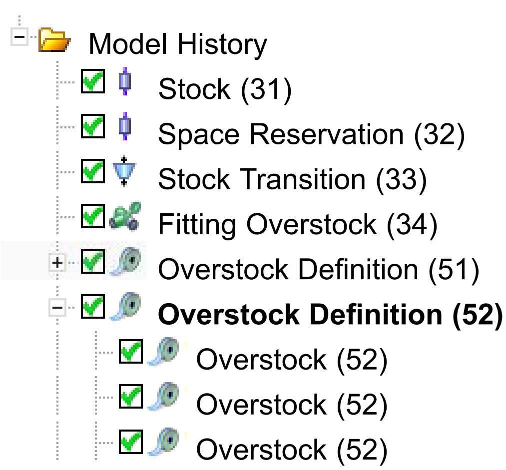
Routing

Stock enhancements

Stock representation

Part Navigator representation

The following are now represented in the **Part Navigator** with distinct icons and labels.



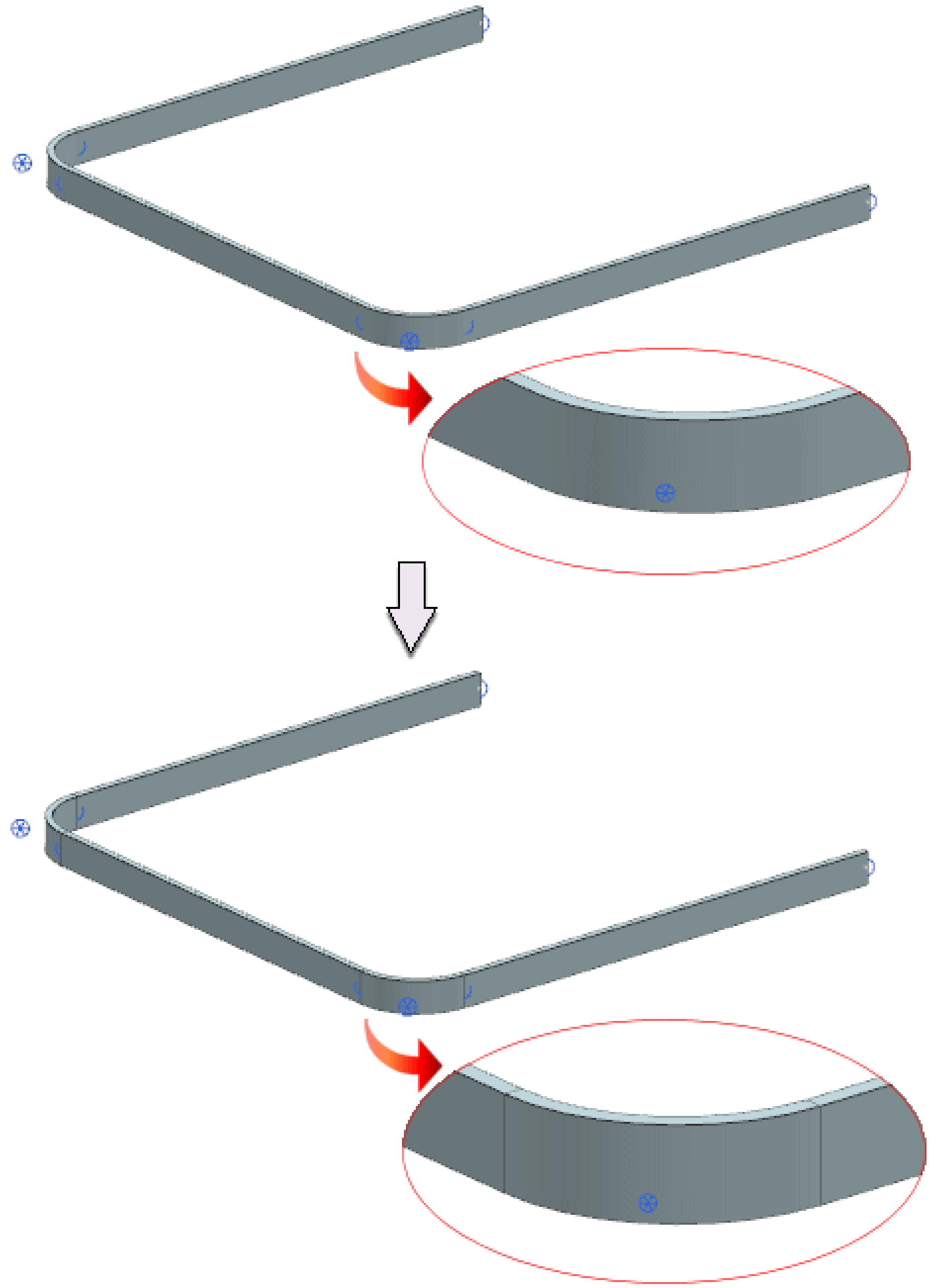
- **Stock**
- **Space Reservation**
- **Stock Transition**
- **Fitting Overstock**
- **Overstock**

Note

- **Fitting Overstock** is available only in Routing Electrical.
- Overstock that is defined together is grouped under an expandable node labeled **Overstock Definition**. If you are using Stock as Components, **Overstock Definition** is a single node in the **Part Navigator**.

HVAC stock representation in the graphics window

When you create non-circular HVAC stock, NX represents the bend segments on the stock as lines on the face of the stock. This creates a more realistic representation of manufacturable stock.



Note

To create this appearance in stock created in pre-NX 10 versions, edit the stock and use the **Split body faces when arc segments are present** ☑ check box.

Why should I use it?

With the new representations in the **Part Navigator** you can:

- Use standard **Part Navigator** functionality to work with stock. For example, the easiest way to edit a feature is by double-clicking it in the **Part Navigator**.

- Easily differentiate between the different types of stock created in the design.

Where do I find it?

Part Navigator representations

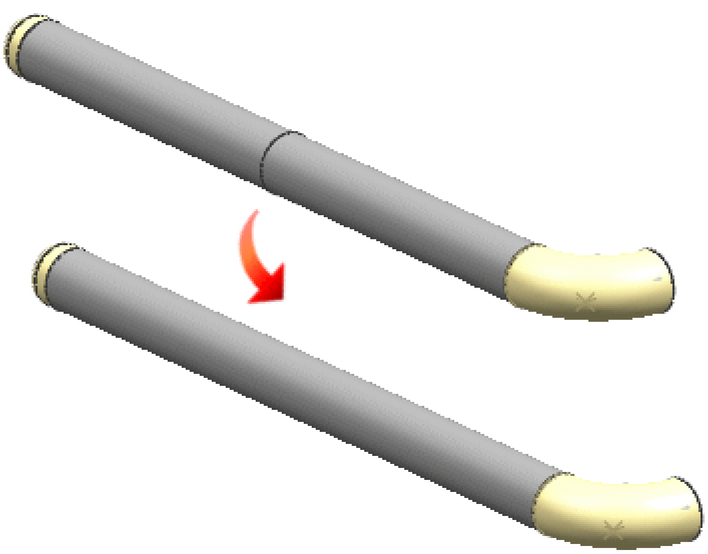
Application	**Routing**
Prerequisite	You must create stock, overstock, stock transition, space reservation, or fitting overstock to see it in the **Part Navigator**.

Split body faces when arc segments are present check box.

Application	**Routing**
Command	**Edit Stock**
Location in dialog box	**Edit Stock** dialog box→**Settings** group→**Split body faces when arc segments are present** check box

Merge Stocks

Use this command to manually merge the stock or space reservation on two or more segments.



NX automatically merges stock only when you use the **Assign Corner** and **Remove Discontinuity** commands.

Note

You cannot merge stock that are not connected, or stock that have different properties or attributes such as material, NPS, or display color.

Where do I find it?

Application	**Routing**
Command Finder	**Merge Stocks**

Upgrading pre-NX 10 stock

The data model has changed for the following from NX 10 onwards:

- **Stock**
- **Overstock**
- **Space Reservation**
- **Stock Transition**
- **Fitting Overstock**

  Note

  Fitting Overstock is available only in Routing Electrical.

In NX 10 or later, you must upgrade stock to the new data model in order to use any Routing command on a part or assembly which has stock saved in pre-NX 10 versions.

Batch upgrade

Use the **refile_part** utility to upgrade stock in large assemblies to the new data model. The **refile_part** utility can be used for upgrading both Legacy Stock and Stock as Components.

> **Note**
>
> If stock components from the assembly are missing, then the **refile_part** utility fails. To recreate missing stock components while using the **refile_part** utility to upgrade stocks to the new data model, use the `-recreate_missing_stock_components` option.

Use case	refile_part syntax
Upgrading the stocks in assemblies while using Teamcenter Integration	ugmanager_refile_program -u=<username> -p=<password> -rec
Upgrading the stocks in assemblies while using native NX	refile_part.exe -y -recreate_missing_stock_components -d
Upgrading the stocks in a single harness assembly part file while using native NX	refile_part.exe -y -recreate_missing_stock_components <fo

Upgrading assemblies with Legacy Stock

When you open a part or assembly which has Legacy Stock saved in pre-NX 10 versions in NX 10, NX automatically upgrades the Legacy Stock so that it is compatible with the new data model. This upgrade happens during part loading when you open the part or assembly and save the part or assembly after it loads. Subsequent attempts to open and load the part or assembly are faster as the stock is upgraded only once.

If you want to defer the stock upgrade, use the **Defer Legacy Stock Conversion During Part Load** customer default. If you defer the upgrade, you must use the **Upgrade Stocks** command to upgrade the stock before you make any changes to the design.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default**.

Upgrading assemblies with Stock as Components

For stock saved in pre-NX 10 versions as Stock as Components, use the **Upgrade Stocks** command to upgrade the stock to the new data model. When you use this command, you can also recreate any stock components that are missing and cannot be loaded.

Where do I find it?

Application	**Routing**
Command Finder	**Upgrade Stocks**
Command line input	**refile_part**

Recreate Inaccessible Stock Components

Use this command to recreate the following components if they are missing and cannot be loaded in the assembly:

- **Stock**
- **Overstock**
- **Space Reservation**
- **Stock Transition**
- **Fitting Overstock**

Note

Fitting Overstock is available only in Routing Electrical.

Where do I find it?

Application	**Routing**
Prerequisite	All stock in the part or assembly must be converted to the new data model.
Command Finder	**Recreate Inaccessible Stock Components**

Loading stock components

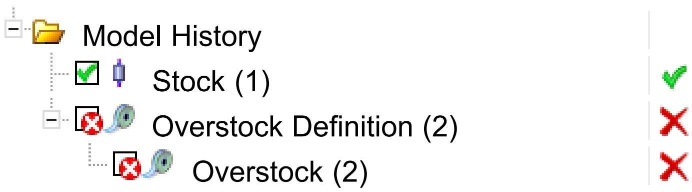
When you load an assembly that has Stock as Component type of stock, NX does not load the stock components automatically. You must fully load all stock components to avoid inconsistent results.

If you partially load an assembly, NX loads the stock components but not the sweep features of individual stock. In such cases, when you change the segment to which a stock is associated, for example, change its length or add parts to the segment, the sweep features of the stock are not updated.

Overstock enhancement

What is it?

When overstock fails, NX no longer deletes it. Failed overstock is now displayed in the **Part Navigator**. Double-click the overstock node in the **Part Navigator** to edit and fix the overstock.



Where do I find it?

Application	Routing
Part Navigator	Double-click a node that represents failed overstock.

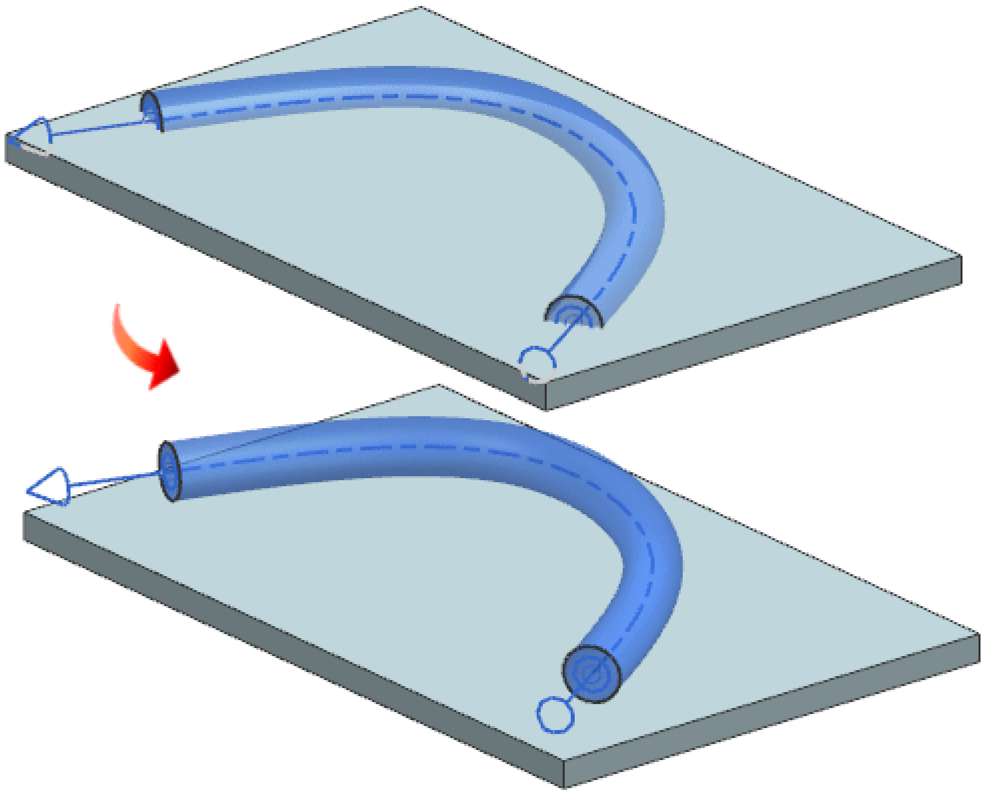
Spline Path enhancement

What is it?

You can now accurately preview a spline path that lies on a stock offset point, a stock offset port, or a stock offset plane. To do this, use the **Show Result** option.

Example

The example shows a spline path that is created using a stock offset point. When you click **Show Result** , NX updates the preview to accommodate the stock offset point.



Why should I use it?

The **Preview** check box creates a dynamic preview that changes as you make selections and set parameters in the **Spline Path** dialog box. It does not however take into account stock offset points, stock offset ports, or stock offset planes that lie on the spline path. If you want NX to take these objects into account, click **Show Result**.

Tip

To improve performance, clear the **Preview** check box and use the **Show Result** option only when you want to preview the final spline path.

Where do I find it?

Application	Routing
Command Finder	**Spline Path**
Location in dialog box	**Preview** group→**Show Result**

Pipe Welding commands

The following commands are now available in the Routing application. These commands work in the same way as they do in Structure Welding. Use them in Routing to work with welding joints on pipes.

Assign Weld Attributes

Use this command to assign weld attributes to curves or edges and edit them.

Export Welding Joints

Use this command to simultaneously export multiple welding joints in the design.

Fabrication Label

Use this command to simultaneously apply labels and annotations on multiple welding joints.

Fabrication PMI

Use this command to simultaneously apply PMI symbols on multiple welding joints.

Use these commands to reduce user-interaction while working with welding joints in Routing.

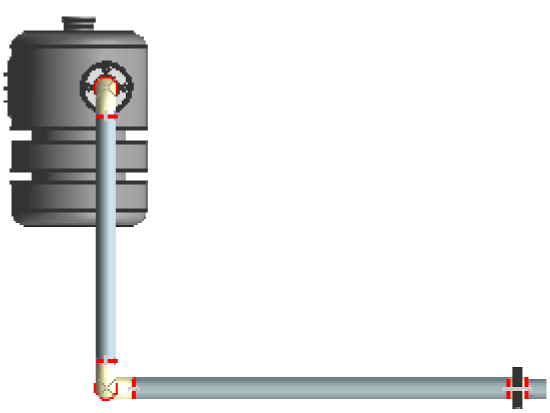
Where do I find it?

Application	**Routing Mechanical→More→Pipe Welding**
Command Finder	**Assign Weld Attributes** **Export Welding Joints** **Fabrication Label** **Fabrication PMI**

Pipe Welding Joint

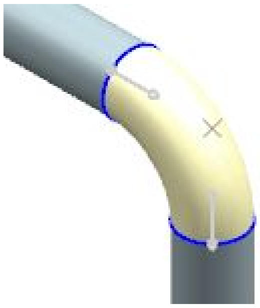
Use this command to create different types of pipe welding joints.

The illustration shows a run with pipe welding joints shown in red.

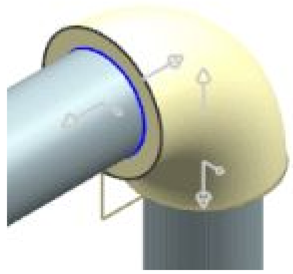


You can create each joint individually or create multiple welding joints simultaneously.

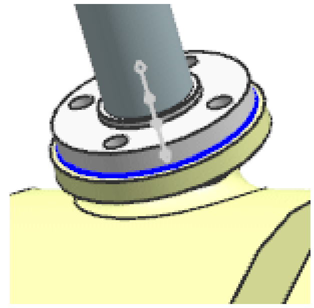
The illustration shows the types of welding joints you can create. The welding joints are shown in blue.



Butt

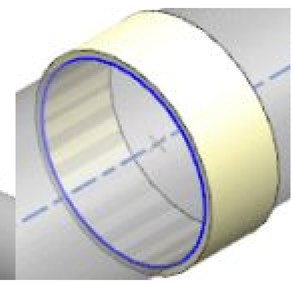


Socket



Mechanical

Boss



Sleeve

Where do I find it?

Application	**Routing Mechanical** (**Pipe Welding**)
Command Finder	**Pipe Welding Joint**

Transform Path enhancements

Inheriting attributes

When you move or copy routing objects using the **Transform Path** command, you can now specify a separate routing object from which you want the moved or copied object to inherit attributes. To do this, use the **Select Reference Object** option.

For example, if you copy a pipe from one run to another run, you can specify a pump on the second run from which the copied pipe inherits attributes. Examples of attributes that can be inherited include operating pressure and operating temperature.

If you do not select a reference object:

- A moved object retains its attributes.
- A copied object inherits the attributes of the object of which it is a copy.

Inheriting the value of an attribute

You can specify the object from which NX inherits an attribute, or specify if you want NX to ignore the attribute. To do this, use the following options.

Source
Inherits the value of the attribute from the object that you move or copy.

Destination
Inherits the value of the attribute from the reference object that you specify.

Omitted
Ignores the attribute for the object that you move or copy.

Check for Duplicate Segments

The **Check for Duplicates** option is now called **Check for Duplicate Segments**. When you select this check box NX detects duplicate segments, and displays the **Resolve Duplicates** dialog box.

Where do I find it?

Application	Routing
Command Finder	**Transform Path**
Location in dialog box	• **Copy Attributes** group→**Select Reference Object** • **Copy Attributes** group→**Copied From** column→Right-click the cell for the attribute whose value you want to change→**Source** or **Destination** or **Omitted** • **Settings** group→**Check for Duplicate Segments**

Placing connecting parts on flanges

Placing connecting parts on flanges

You can now easily connect parts to a pair of flanges when you place the flanges on stock. Use connecting parts such as a gasket, studs, bolts, and nuts. Setting up the placement of connecting parts on flanges makes downstream tasks easier. For example, during or after the completion of the design you can easily keep track of the number of connecting parts used in the design because these connecting parts are created as User Defined Objects (UDOs). You can keep track of the connecting parts using the Routing Bill of Materials (BOM). Note that UDOs are not visible in the design or in the **Assembly Navigator**.

When you place a flange to connect it to an existing flange, or when you place a pair of connected flanges using generic post placement, NX displays successive dialog boxes that list the connecting parts chosen for post placement.

The displayed connecting parts are filtered based on the value of an attribute, such as `BOLT_DIA`, which you must specify before you start placing parts. For example, if you choose to place studs, you must set up the specifications so that the studs made available for placement are filtered using the `BOLT_DIA` attribute. NX matches the value of the bolt diameter of studs with the value of the same attribute on the flanges.

The list of results for bolts or studs is filtered further based on the value of the `BOLT_LEN` attribute. NX calculates the minimum required bolt length using information on the flanges and the gasket, if selected, including any user-specified length adjustment. The bolts or studs that are listed have the shortest length that meets this calculation of minimum length. The calculations and the minimum required bolt length are also displayed in the selection dialog box for bolts or studs.

To enable filtering of the connecting parts for placement on other parts such as a globe valve, you must add an attribute, such as the `BOLT_DIA` attribute, to the part table file (PTB file) of the family of globe valve parts, and create a specification using the **Connection Post Placement** dialog box to filter the parts by the specified attribute.

After you place the connected parts in the design, they appear in the Routing BOM.

Note

- Connected parts appear in the Routing BOM, only if you select the options in the **Compatibility Table** dialog box→**Report in BOM** group.

- Generic post placement works only when you place a part on stock.

Example

Workflow for placing bolts and nuts on a valve part that is placed on stock.

In the PTB file for globe valves, add the attribute by which to filter bolts and nuts, for example, `BOLT_DIA`.

↓

Create a new set of specifications:

- Use the **Generic Post Placement** dialog box to place flanges when you place a globe valve.
- Use the **Connection Post Placement** dialog box to add bolts and nuts as connecting parts on the flanges. Filter the bolts and nuts using your attribute.

↓

In the **Compatibility Table** dialog box, in the **Report in BOM** group, make sure **Bolts** ☑ and **Nuts** ☑ are selected.

↓

Set the new specification as the active specification.

↓

Place a globe valve part on stock.

↓

NX places previously defined flanges on the globe valve part.

Two **Connection Post Placement** dialog boxes display connecting parts filtered by your attribute.

Choose one type of bolt and one type of nut to place on the valve part. NX calculates the required length of the bolts, and places the required number of bolts and nuts. Because they are not modeled parts, they do not appear in the assembly.

↓

Create a Routing BOM and check the rows to verify the placement of bolts and nuts and the related information.

Part placement enhancements

Maintaining pipe length

NX cuts the pipe to accommodate the flange that you place on it and maintains the overall length of the pipe when you place the flange in the following conditions:

- You place a flange that has two parallel ports facing opposite directions, such as a weld neck flange or a socket weld flange.
- You place the flange at the end of the pipe.

  In earlier versions, NX maintained the overall length of the pipe only if you placed flanges in the middle of a pipe.

  Note

  Blind flanges have a single port. When you place a blind flange at the end of a pipe, it adds to the length of the pipe.

Placement solutions

The sequence in which NX provides placement solutions is improved. The sequence is based on the following:

- The PREFERRED_PORT attribute.
- The type of port available on the part—Fitting, Fixture, or Multi—changes the order of placement depending on:
  - Whether you place the part in Routing Mechanical, Routing Electrical, or Routing Logical.
  - The object on which you place the part.
- The CROSS_SECTION attribute and the CONNECTION_TYPE attribute when the part is placed on a Fitting port.
- The Value of the FLOW_DIRECTION attribute of the port. The value of this attribute is defined when you qualify a port using the **Qualify Part** dialog box. The part may be placed in one of the following conditions:
  - The object on which you place it is a part of a run. The run must have a defined direction of flow and the part you place must have a defined FLOW DIRECTION property.
  - You place the part on a Fitting port with a FLOW DIRECTION property.

Anchor-based placement solutions

When you use an anchor-based solution to place a part, the Status line displays the name of the anchor that NX uses for the part placement. To set this up, you must specify a name to the anchor while qualifying the part. For example, assume that you are qualifying a part to use an anchor-based solution for placement, and you specify the name of the anchor as **ANCH_1234**. When you place this part, the Status line displays **Anchor: ANCH_1234**.

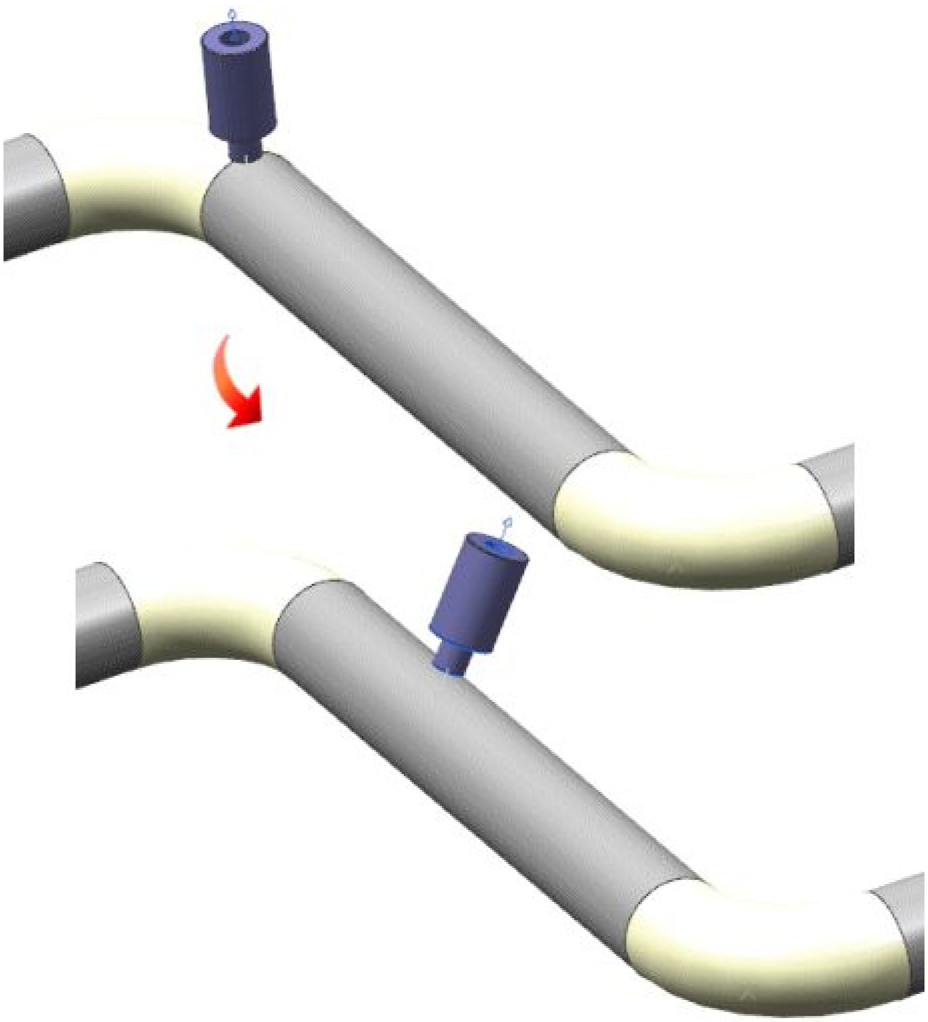
If, while qualifying a part with an anchor-based solution, you retain the default anchor name, then NX displays **:Anchor:** in the **Status** line, when you place that part.

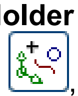
Where do I find it?

Application	Routing
Command Finder	**Place Part** 

Moving a measurement holder

You can change the location and angle of the measurement holders in your design.



To do this, use the **Edit Measurement Holder** dialog box. To open this dialog box, in the **Move Part** dialog box, click **Placement Object** , select the measurement holder that you want to move, and click **OK**.

Note

You can move only those measurement holders that are placed in NX 10.

Where do I find it?

Application	**Routing**
Command Finder	**Move Part**
Location in dialog box	**Placement Object** →Select the measurement holder you want to move→**OK**→**Edit Measurement Holder** dialog box

Sheet Metal

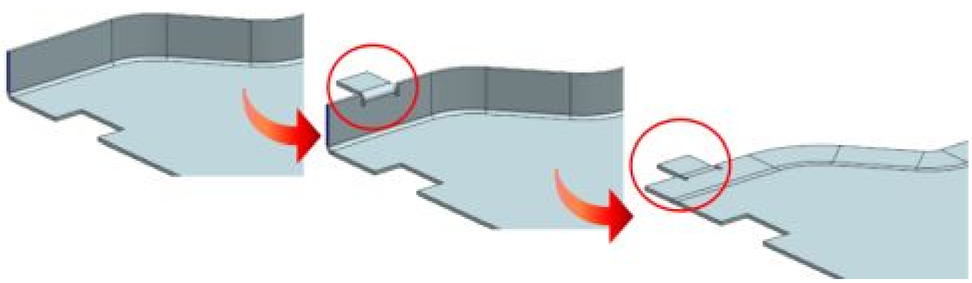
Flange enhancement

What is it?

You can now create a flange on the linear edge of the following:

- A contour flange that is created using the chaining method where the connected curves include a curved edge.
- A hem flange that is created using connected curves that include a curved edge.

For this example, a flange is created on the linear edge of a contour flange, and then both flanges are flattened.



You can:

- Unbend and rebend these flanges.
- View the flattened flange in the flat solid representation or the Flat Pattern view.
- Modify the bend radius, bend angle, or neutral factor values of these flanges using the **Resize Bend Radius**, **Resize Bend Angle**, or **Resize Neutral Factor** command.
- Create tapers on these flanges using the **Bend Taper** command.

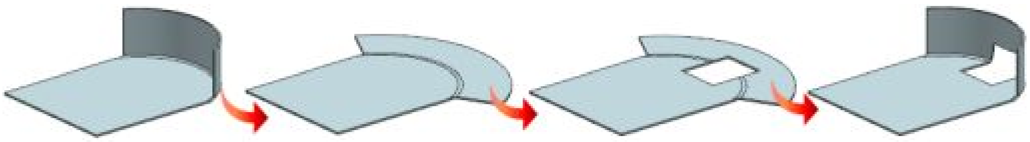
Where do I find it?

Application	Sheet Metal
Command Finder	**Flange**

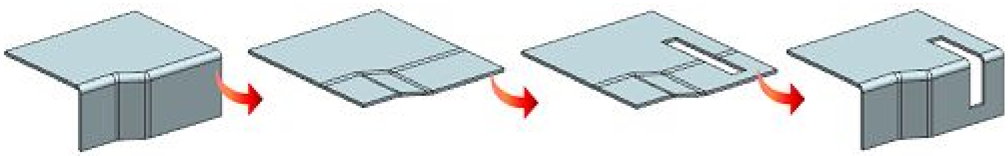
Unbend and Rebend enhancements

What is it?

You can now use the **Unbend** and **Rebend** commands to unform and reform advanced flanges and joggles. You must first convert the advanced flanges and joggles to sheet metal using the **Convert to Sheet Metal** command.



Advanced flange



Joggle

Where do I find it?

Application	Sheet Metal
Command Finder	**Unbend** **Rebend**

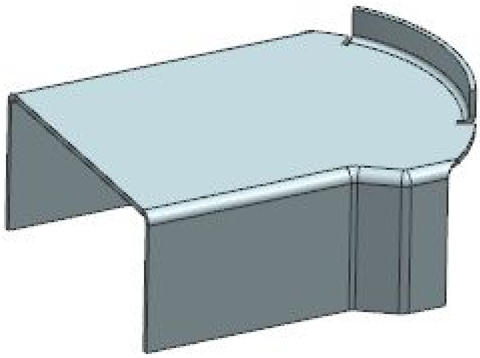
Convert to Sheet Metal enhancements

What is it?

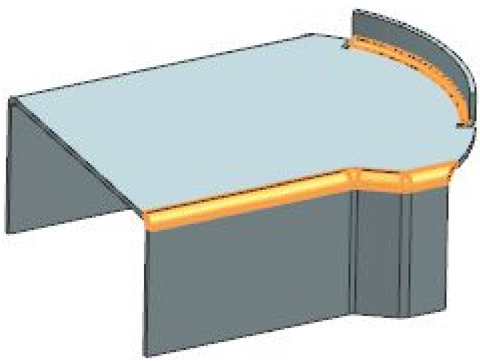
When you convert a non-sheet metal part to sheet metal, you can now select bend faces of advanced flanges and joggles.

After the conversion, you can use the **Unbend** command to flatten the part, or use the **Flat Pattern** command to create the flat pattern view.

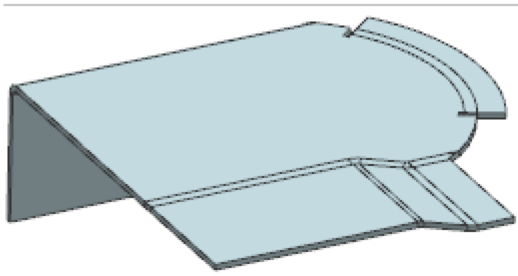
The example shows the workflow for unbending and rebending an advanced flange and a joggle.



Original sheet metal part



Use the **Convert to Sheet Metal** command to convert the bend faces of the advanced flange and joggle to sheet metal. To select the bend faces that you want to convert, use the options in the **Additional Geometry** group.



After conversion, use the **Unbend** command to flatten the advanced flange and joggle.

Where do I find it?

Application	Sheet Metal
Command Finder	**Convert to Sheet Metal**
Location in dialog box	**Additional Geometry** group

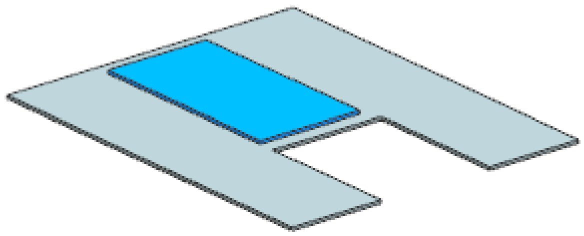
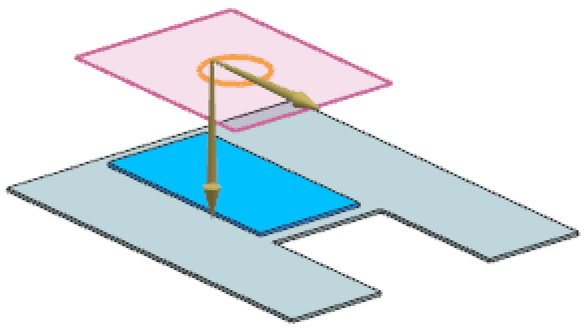
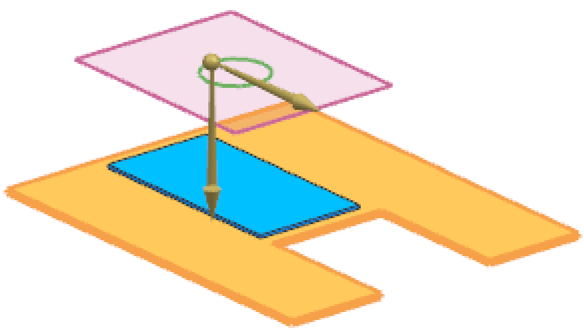
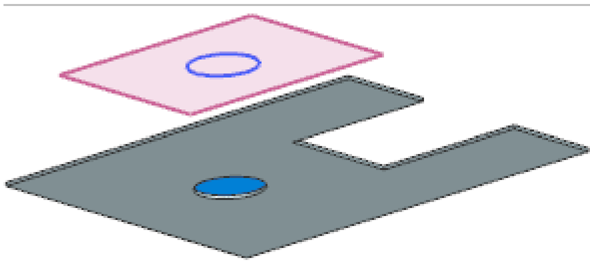
Specifying a target body for creating normal cutouts

What is it?

You can now specify the target body on which you want to create the normal cutout feature.

If the part has a single sheet metal body, NX infers it as the target body.

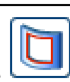
The example shows the workflow for creating a normal cutout on a target body in a part that has multiple sheet metal bodies.

	Original sheet metal part
	Specify the normal cutout section. For this example, **Limits** = **Through All**.
	Select the target body.
	NX creates the cutout on the specified target body.

Why should I use it?

Specifying the target body to create a cutout is useful when the part has multiple sheet metal bodies.

Where do I find it?

Application	Sheet Metal and Flexible Printed Circuit Design
Command Finder	**Normal Cutout** 
Location in dialog box	**Target** group→**Select Body** option

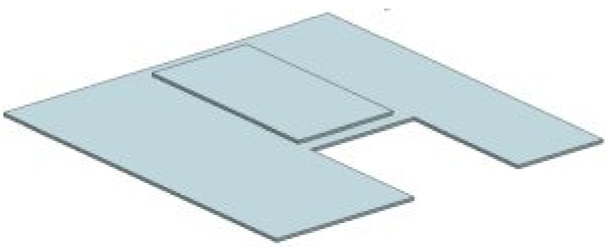
Specifying a target face for creating bends and jogs

What is it?

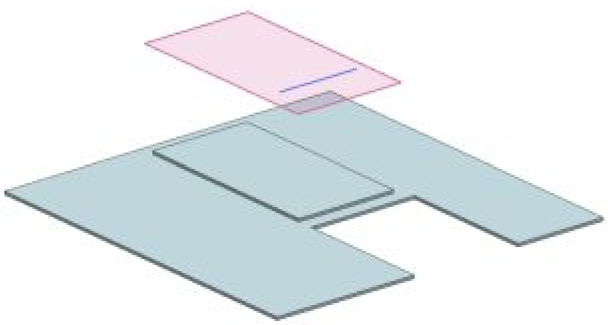
You can now specify the target face on which you want to create a bend or jog feature.

If the bend line lies on a face of a sheet metal body, NX infers that face as the target face. You can specify a different face as the target face if required.

The example shows the workflow for creating a bend on a target face in a part that has multiple sheet metal bodies.

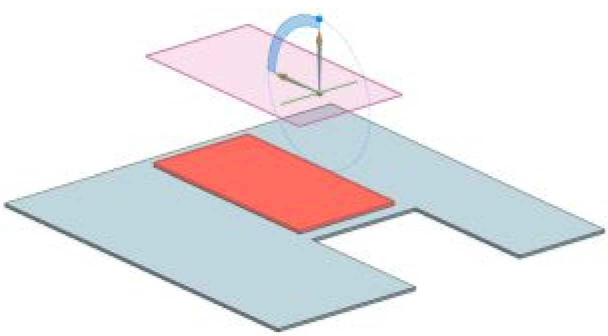


Original sheet metal part

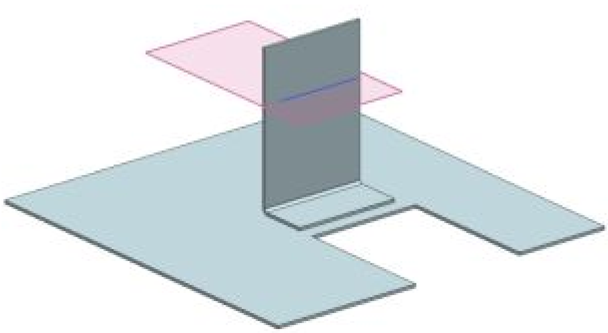


Specify the bend line.

The bend line must lie on a plane that is parallel to the target face.



Select the target face.



NX creates the bend on the body of the specified target face.

Why should I use it?

Specifying a target face to create a bend or a jog is useful when the part has multiple sheet metal bodies.

Where do I find it?

Application	Sheet Metal and Flexible Printed Circuit Design
Command Finder	**Bend**  **Jog** 
Location in dialog box	**Target** group→**Select Face** option

Gusset enhancements

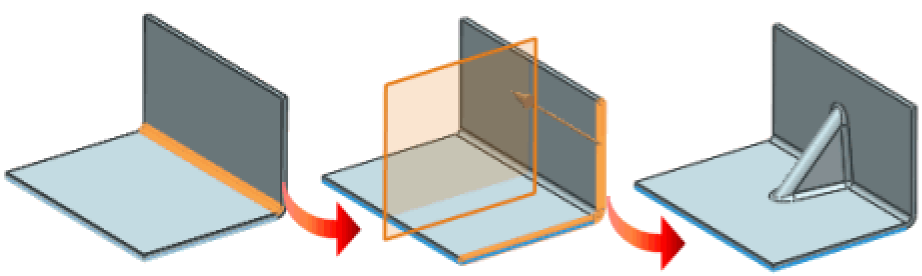
What is it?

The existing **Gusset** command is enhanced.

Creating gusset features

You can now specify the location of gussets using a datum plane as reference.

To do this, use the **Specify Plane** options in the **Gusset** dialog box. The datum plane must be perpendicular to the axis of the bend.

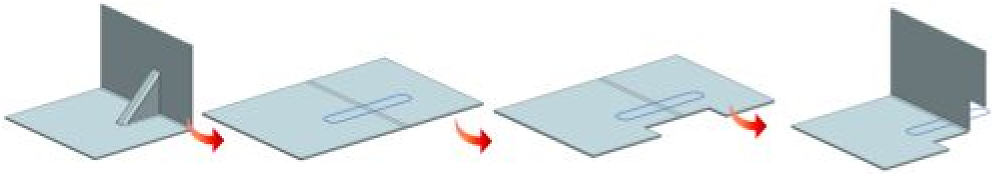


Previously, the location of gussets was specified using the start edge as reference. If the start edge was modified upstream, the gusset would fail.

NX 10 onwards since you use datum planes instead of edges to specify the location of gussets, you ensure that the gusset does not fail when you perform operations upstream that affect the start edge. If you reorder the gusset feature in the **Part Navigator**, for example to re-create a gusset after an unbend-rebend operation, there is a greater chance that gussets are successfully re-created.

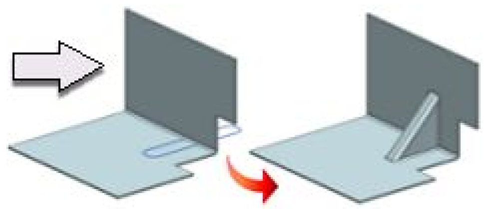
Example

In the example, the flange is unbent to create a normal cutout and then rebent. Notice that NX removes the gusset feature after the unbend operation.

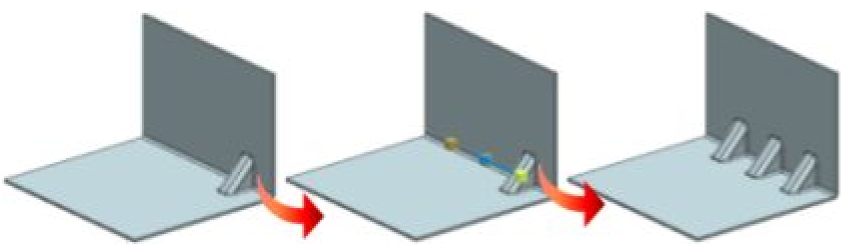


To retain the gusset after the unbend operation, in the **Part Navigator**, reorder the gusset feature after the Rebend feature to re-create the gusset.

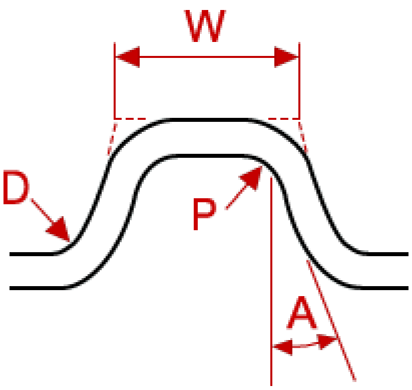
- Model History
  - Datum Coordinate System (0)
  - SB Tab (1)
  - SB Flange (2)
  - SB Unbend (3)
  - Datum Plane (4)
  - SB Normal Cutout (5)
  - SB Rebend (6)
  - SB Gusset (7)



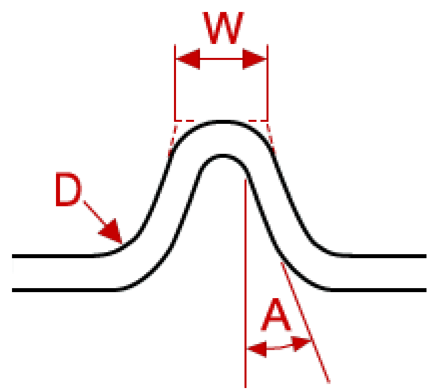
The **Gusset Placement** group in the **Gusset** dialog box is removed. To create multiple gussets on the same bend, use the **Pattern Feature** command.



The **Corner Radius** option is removed. For square gussets, you can specify the **Die Radius** (D) and **Punch Radius** (P) values separately. For round gussets, you can specify the **Die Radius** (D) value. The resulting gussets will now have uniform thickness.



Square gusset



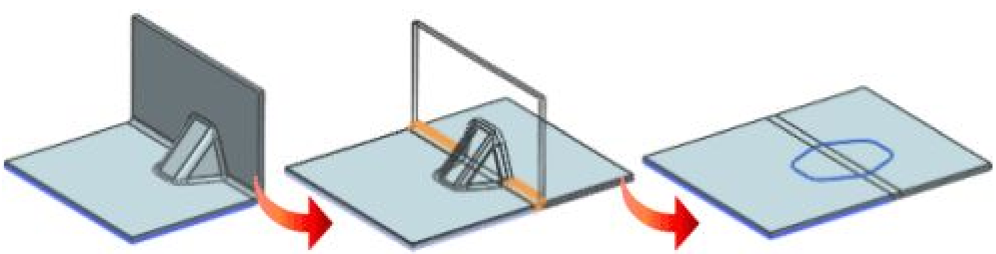
Round gusset

While creating gussets you can switch between the **Automatic Profile** and the **User Defined Profile** type. In this case, all dialog box inputs are retained, until you click **OK** or **Cancel** to close the **Gusset** dialog box.

Flattening bends with gusset features

When you use the **Unbend** command to flatten a bend across which a gusset feature exists, NX now displays an alert which indicates that the gusset feature will be removed during the unbend operation.

You can now specify if you want to automatically extract the boundary curves of the gusset and display them on both layers in the flattened sheet metal part. To do this, select the **Extract Gusset Curves** check box in the **Unbend** dialog box.



Customer defaults

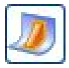
The gusset customer defaults are modified. The **Rounding** tab is renamed to **Settings**.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default**.

Where do I find it?

Application	Sheet Metal

Command Finder	**Gusset**  **Unbend** 
Location in dialog box	**Gusset** dialog box→**Location** group→**Specify Plane** options **Unbend** dialog box→**Additional Curves or Points** group→**Extract Gusset Curves**  check box

Bend Taper enhancements

What is it?

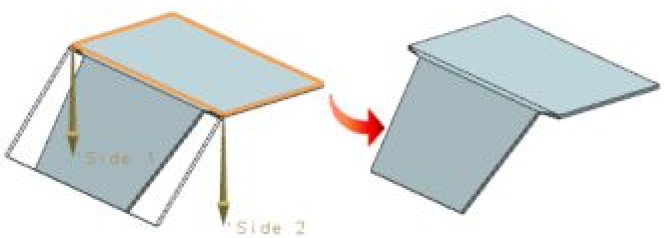
You can now add or remove material from the bend and web to which you apply a taper. You can also create reverse tapers as NX now supports negative values for the bend taper distance or angle, and web taper angles. A dynamic preview of the bend taper is now available.

Specifying stationary faces or edges

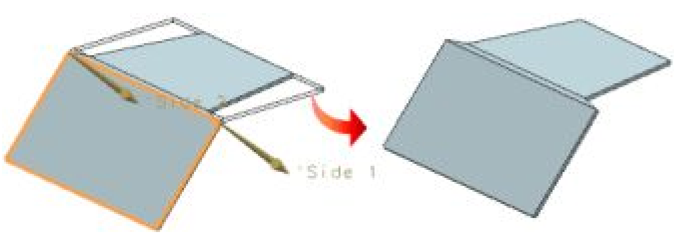
You can specify a face or an edge that remains stationary during the taper operation. NX uses the selected face or edge to infer the side that remains fixed during the taper operation.

Example

In the following example, the top face is stationary.



In the following example, the bottom face is stationary.



To select a stationary face or edge, use the **Select Face or Edge** option.

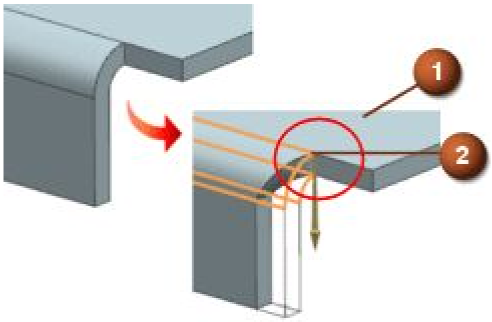
Specifying a start location for the taper

You can specify if you want to create tapers that start from the bend or the web.

To do this, use the following options:

Taper from Bend

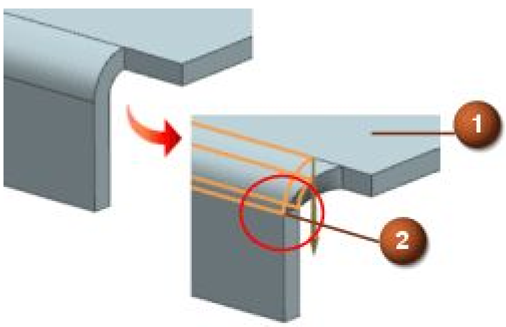
NX adjusts the width of the web and creates a taper that starts from the bend.



1	Stationary face
2	Start point of the taper

Taper from Web

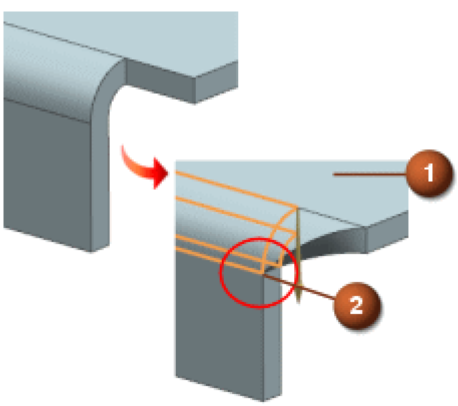
NX maintains the width of the web and creates a taper that starts from the web.



1	Stationary face
2	Start point of the taper

You can create bend tapers that extend up to the end of the base flange or the tab.

To do this, use the **To End** option.



1	Stationary face
2	Start point of the taper

Creating reverse tapers

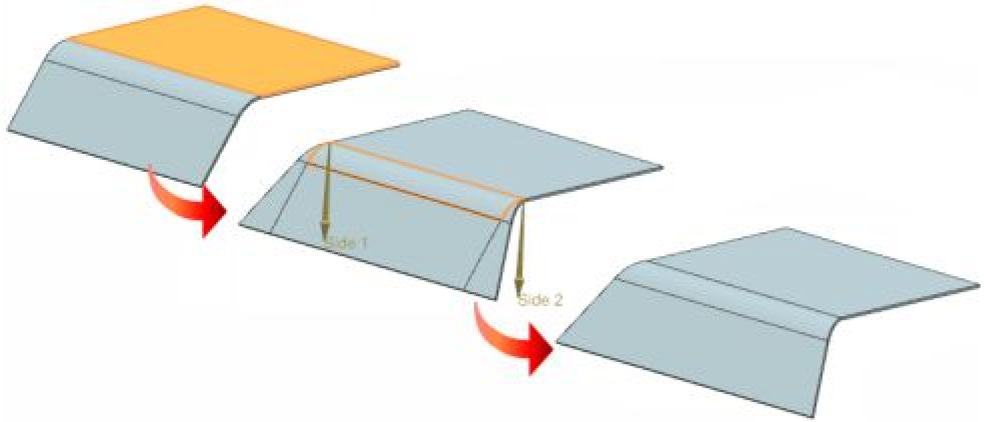
You can create tapers by adding material to the flange.

To do this:

- If you select **Taper from Bend** you must enter negative values for both the taper distance or taper angle of the bend, and the taper angle of the web.

- If you select **Taper from Web** you must enter a positive value for the taper distance or taper angle of the bend, and a negative value for the taper angle of the web.

In the example, a symmetric taper that starts at the bend is created with the taper angle for the bend and the web set to **–20**.



Where do I find it?

Application	Sheet Metal and Flexible Printed Circuit Design
Prerequisite	The **To End** option in the **Input Method** list is available only when you select the **Taper from Web** option in the **Taper Definition/Taper Definition Side 1/Taper Definition Side 2** group.
Command Finder	**Bend Taper**
Location in dialog box	• **Stationary Face or Edge** group→**Select Face or Edge** • **Taper Definition/Taper Definition Side 1/Taper Definition Side 2** group→**Bend** subgroup→**Taper from Bend** or **Taper from Web** • **Taper Definition/Taper Definition Side 1/Taper Definition Side 2** group→**Bend** subgroup→**Input Method** list→**To End** • **Taper Definition/Taper Definition Side 1/ Taper Definition Side 2** group→**Bend** subgroup→**Taper Distance** or **Taper Angle** • **Taper Definition/Taper Definition Side 1/ Taper Definition Side 2** group→**Web** subgroup→**Taper Angle**

Flexible Printed Circuit Design

Flat Solid enhancements

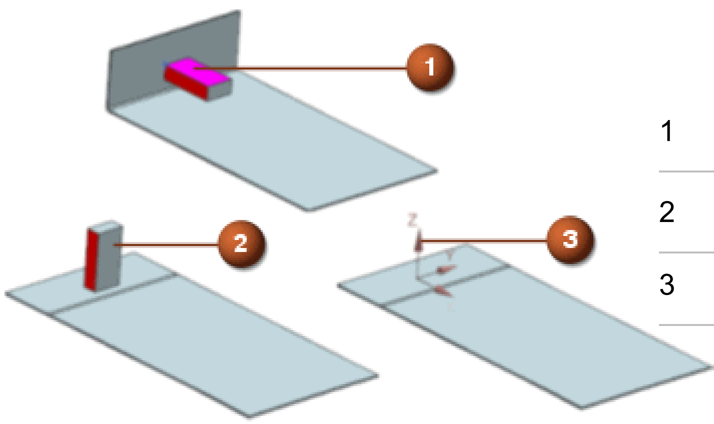
What is it?

You can now represent PCB components and restriction areas on the flat solid of an FPCD part.

Representing components on the flat solid

NX creates a copy of the PCB component, replicates the PCB Exchange attributes, and transforms the PCB component on the flat solid. The replicated attributes are non-associative.

To indicate the location of a PCB component on the flat solid, you can represent it as a solid body or a coordinate system.



1	PCB component in the FPCD assembly
2	PCB component represented as a body
3	PCB component represented as a CSYS

Note

To orient the PCB component correctly on the flat solid you must ensure that the Z-axis of the PCB component is normal to the placement face of the FPCD part.

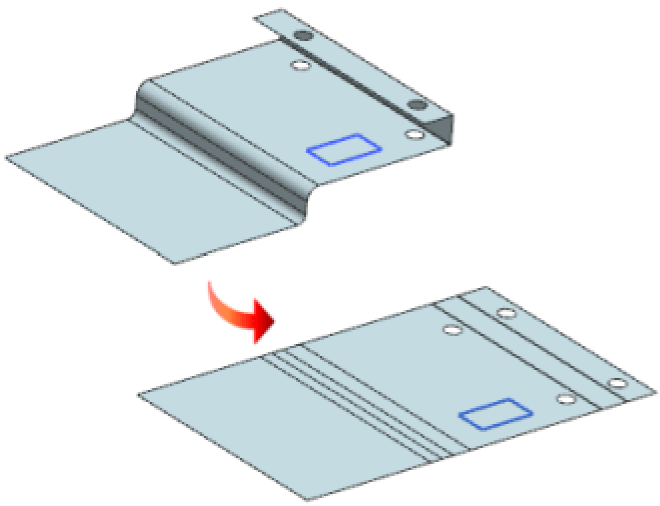
If you create a PCB component as a body within the native FPCD part, and you want to represent it on the flat solid, you must group the PCB component with the associated datum CSYS.

Representing restriction areas on the flat solid

You must define the restriction areas in PCB Exchange. NX represents the restriction areas as sketches on the flat solid.

A restriction area specified as a solid body or a sheet body must have a sketch associated with it.

- If the restriction area has an internal sketch associated with it, NX uses that sketch as a reference to represent it on the flat solid.
- If the restriction area has an external sketch associated with it, you must group the external sketch and the body on which you create the external sketch.



Note

To orient the PCB restriction area correctly on the flat solid, you must ensure that the sketch that represents the restriction area lies on the placement face of the FPCD part.

Why should I use it?

These enhancements help you to visualize the location of PCB components and restriction areas of an FPCD part in the flattened state.

Where do I find it?

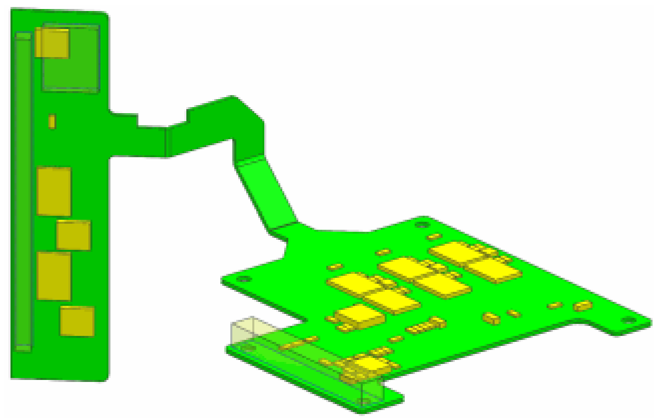
Application	Flexible Printed Circuit Design
Command Finder	**Flat Solid**
Location in dialog box	**Settings** group→**Transform Components** list→**None** or **Body** or **CSYS** **Settings** group→**Transform Restriction Areas** check box

PCB Exchange

Support of a Flat Solid version of a flexible PCB

What is it?

You can now import and export ECAD models that are in their bent state as long as the `NxFlexFlatSolid` variable is set to `Yes` in the *pcbx_ug_model.ini* file. Previously, the import and export was restricted to the flattened 2D state of the board.



Why should I use it?

You can now work using the full 3D representation of a flexible PCB in PCB Exchange.

Where do I find it?

PCB Exchange looks for *pcbx_ug_model.ini* in the following locations:

1. Network location specified by the `MAYA_PCB_ENV_DIR` environment variable.

2. Working directory from where you launch PCB Exchange which is typically the same directory from which you launch NX.

3. Location specified by the `MAYA_PCB_DIR` environment variable.

4. The NX installation directory (*%UGII_BASE_DIR%\UGPCBXCHANGE*).

Multi-Height Area Creation

What is it?

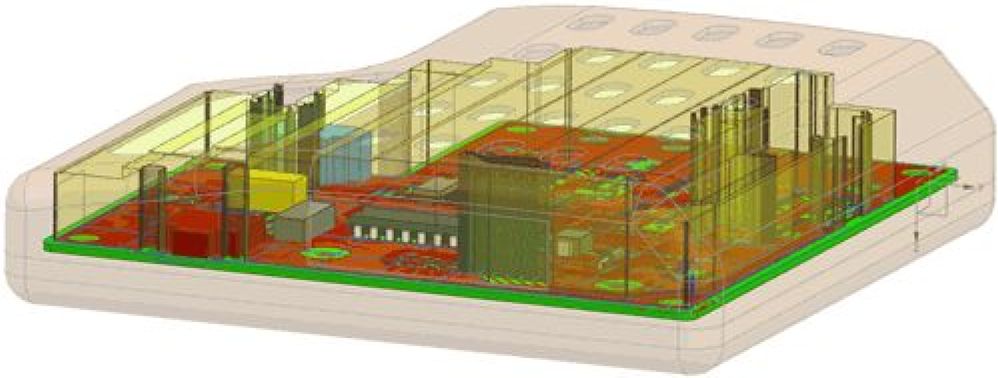
Use the new **Multi-Height Area Creation** command to create restriction areas that are bounded by polygon faces. You specify:

- The face on which the restriction area will be placed.

- One or more bounding objects.

- Resolutions along the X and Y axes.

- Restriction area attributes, such as type, name, owner, and color.



PCB Exchange creates extruded rectangular bodies of resolution X and Y sizes and height going up to the bounding object. It then unites all bodies and assigns the restriction area attributes to the united body. You can choose not to unite all the bodies. In that case, PCB Exchange assigns the restriction area attributes to each body.

Why should I use it?

This command lets you create restriction areas with variable height that are bounded by other objects in the assembly.

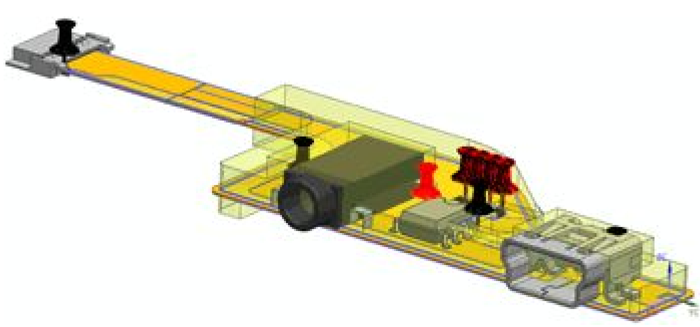
Where do I find it?

Application	PCB Exchange
Command Finder	**Multi-Height Area Creation** 

Markers stored as faceted geometry

What is it?

Markers are now stored as faceted geometry in .stl files that are stored in the *%UGII_BASE_DIR%\UGPCBXCHANGE\data* folder. By default, NX uses the faceted geometry stored in *mark.stl* to display markers as a pushpin.



You can use other shapes for your markers. You specify the name of the files in the following variables in the *pcbx_ug_model.ini* file.

Variable	Value options	Specifies the marker file name for displaying
`NxPinMarkSTLName`	`"[name of STL file without extension]"` `"mark"` (default)	ECAD primary pins. They are always displayed in red.
`NxMCADPinMarkSTLName`		MCAD primary pins. They are always displayed in black.
`NxPadMarkSTLName`		ECAD pads. They are always displayed in green.
`NxTestPointMarkSTLName`		Test points. They are always displayed in blue.

In previous versions, the displayed markers were NX features stored in part files. If you want to use NX part files as markers instead of faceted geometry, set the following in the *pcbx_ug.ini* file:

```
PCBxDisplayOldMarkers = yes
```

Why should I use it?

Using faceted geometry instead of NX features to store markers speeds up the display time when you have a large number of markers.

Where do I find it?

PCB Exchange looks for *pcbx_ug_model.ini* and *pcbx_ug.ini* in the following locations:

1. Network location specified by the `MAYA_PCB_ENV_DIR` environment variable.
2. Working directory from where you launch PCB Exchange which is typically the same directory from which you launch NX.
3. Location specified by the `MAYA_PCB_DIR` environment variable.
4. The NX installation directory (*%UGII_BASE_DIR%\UGPCBXCHANGE*).

Visualize Primary Pins

What is it?

Use the new **Visualize Primary Pins** command to display MCAD and ECAD primary pin markers for the selected components.

In the **Visualize Primary Pins** dialog box, you can:

- Select components from the graphics window or the **Assembly Navigator**.
- Select components that have the same Teamcenter item type defined by the DB_PART_TYPE attribute.
- Display the information regarding the alignment of MCAD and ECAD primary pins in the **Information** window.
- Reset the current selection and display of markers to be able to display markers of other components without exiting the dialog box.

Why should I use it?

This command lets you easily see if the MCAD and ECAD primary pin markers for a selected component are defined, aligned, or mismatched.

Where do I find it?

Application	PCB Exchange
Command Finder	**Visualize Primary Pins**

Primary pin information on components

What is it?

The **PC Design Navigator** command now displays the information regarding MCAD and ECAD primary pin markers for components in the **Primary Pin** column. You can find the following values in the **Primary Pin** column.

Match — Indicates that the MCAD primary pin location matches the ECAD primary pin location for the given component.

Mismatch — Indicates that the MCAD primary pin location does not match with the ECAD primary pin location for the given component.

No ECAD Pin — Indicates the given component does not have an ECAD primary pin defined.

No MCAD Pin — Indicates the given component does not have an MCAD primary pin defined.

In the **PC Design Navigator** dialog box, you can choose to display the MCAD and ECAD primary pin markers on the model.

Why should I use it?

You now have primary pin information for all your components directly in the **PC Design Navigator** dialog box.

Where do I find it?

Application	PCB Exchange
Command Finder	**PC Design Navigator**
Location in dialog box	**Components** tab→right-click a row→**Visualize Primary Pin**

Refresh Primary Pin Mismatch Info

What is it?

Use the new **Refresh Primary Pin Mismatch Info** command to display the *Primary Pin Comparison Report* in the NX **Web Browser**. The report for each component lists the following:

- The package name

- The part number
- The result of the primary pin comparison

Match	Indicates that the MCAD primary pin location matches the ECAD primary pin location for the given component.
Mismatch	Indicates that the MCAD primary pin location does not match with the ECAD primary pin location for the given component.
No ECAD Pin	Indicates the given component does not have an ECAD primary pin defined.
No MCAD Pin	Indicates the given component does not have an MCAD primary pin defined.

Why should I use it?

In the HTML report that you get when you use the **Validate PCA** command, PCB Exchange lists the result of the primary pin comparison at the time you import your ECAD model. The **Refresh Primary Pin Mismatch Info** command lets you view the changes that were done to the primary pin markers in your model after the import.

Where do I find it?

Application	PCB Exchange
Prerequisite	In the **PCB Exchange Settings** dialog box, the **Compare Primary Pin Locations on Import** check box must be selected.
Command Finder	**Refresh Primary Pin Mismatch Info**

Remove Holes with Size filter

What is it?

You can now filter non-circular holes when importing, exporting, or updating, your model, or when creating a thermal solution. NX filters the non-circular hole based on the size of the hole's bounding box. You specify values in **Less than** and **Greater than** boxes to remove holes with sizes that are less than or greater than the values specified.

Why should I use it?

This filter lets you better control the aspect of your NX model with respect to non-circular drilled holes.

Where do I find it?

Application	PCB Exchange

Command Finder	**Import ECAD Model** or **Export ECAD Model** or **Compare and Update PCA** or **Create ESC Solution** or **PCB Exchange Settings**

Import ECAD Model or **Export ECAD Model** commands

Location in dialog box	**Use Entity Filter** →Filters List (**Filters List** dialog box) **Pre-Defined Filters** tab→**Remove Holes with Size**

Compare and Update PCA command

Location in dialog box	**Use NX Filters on NX Model** or **Use NX Filters on ECAD Model** →Filters List (**Filters List** dialog box) **Pre-Defined Filters** tab→**Remove Holes with Size**

Create ESC Solution command

Location in dialog box	**Create Idealized Part** →**Use Entity Filter** →Filters List (**Filters List** dialog box) **Pre-Defined Filters** tab→**Remove Holes with Size**

PCB Exchange Settings command

Location in dialog box	**IDX Collaboration** tab→**Entity Filters on NX Model** or **Entity Filters on ECAD Model** →Filters List (**Filters List** dialog box) **Pre-Defined Filters** tab→**Remove Holes with Size**

Filtering entities by ownership

What is it?

You can now create custom filters to filter entities by ownership when importing, exporting, or updating your model, or when creating a thermal solution. In previous versions, you could only filter components by ownership, now you can also filter drilled holes and restriction areas by ownership. You add custom filters in the *pcbx_ug_filter.ini* file.

Variable	Value options	Specifies filtering by ownership for
KeepinOwner	• ALL (default) • +/-ECAD • +/-MCAD • +/-UNOWNED • NONE	Keep-in areas
KeepoutOwner		Keep-out areas
OtherOwner		Other areas
CopperOwner		Copper areas
DrilledHoleOwner		Drilled holes

For each of the variables, you can define multiple values as a comma delimited list. A plus sign (+) in front of the value indicates that it is to be included. A minus sign (-) indicates that it is to be excluded. If there is no indication, plus (+) is assumed.

Why should I use it?

Filtering restriction areas, holes, and components by ownership improves the collaboration between ECAD and MCAD designers.

Where do I find it?

PCB Exchange looks for the *pcbx_ug_filter.ini* file in the following locations:

1. Network location specified by the `MAYA_PCB_ENV_DIR` environment variable.

2. Working directory from where you launch PCB Exchange which is typically the same directory from which you launch NX.

3. Location specified by the `MAYA_PCB_DIR` environment variable.

4. The NX installation directory (*%UGII_BASE_DIR%\UGPCBXCHANGE*).

Creation of idealized part

What is it?

You can now do the following when you create a thermal solution in PCB Exchange:

- Create an idealized part file in addition to the FEM and Simulation files.

- Filter out certain holes and components before the creation of the idealized part file.

Why should I use it?

This new feature lets you store the modifications within the idealized part file and keep the original master model geometry unchanged.

Where do I find it?

Application	PCB Exchange

Command Finder	**Create ESC Solution**
Location in dialog box	**Output Options** group→**Create Idealized Part** **Output Options** group→**Create Idealized Part** →**Use Entity Filter**

Creating a Space Systems Thermal solution

What is it?

You can now specify the solution type that you want to create for the thermal analysis of your PCB model in a dialog box. You can create a solution of the following types:

- Thermal
- Electronic Systems Cooling
- Space Systems Thermal

In the previous releases, you could specify the Thermal and Electronic Systems Cooling solution types only in the pcbx_thermal.xml file.

As in other solutions, PCB Exchange now creates **PCB Component** simulation objects and a **Printed Circuit Board** simulation object when creating a Space Systems Thermal solution.

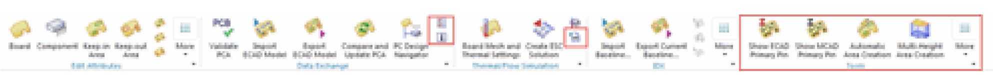
Where do I find it?

Application	PCB Exchange
Command Finder	**Create ESC Solution**
Location in dialog box	**Output Options** group→**Solution Type**

Home tab enhancements

What is it?

You can now access all the PCB Exchange commands on the Ribbon bar from the **Home** tab. In previous releases, most of the commands were only accessible through the **Menu**.



The organization of the commands on the tab also changed. NX now organizes the PCB Exchange commands in the following groups:

- **Edit Attributes**
- **Data Exchange**
- **Thermal/Flow Simulation**

- **IDX**
- **Tools**

PCB Exchange version

What is it?

NX now displays the following information about PCB Exchange:

- The version of PCB Exchange
- The location of the loaded initialization (.ini) files

Why should I use it?

This new feature provides you with general information about PCB Exchange.

Where do I find it?

Application	PCB Exchange
Command Finder	**About PCB Exchange**

Importing holes as curves or cutouts

What is it?

You can now import holes of a certain type as curves or cutouts. In previous versions, you could import all holes only as simple hole features, curves, or cutouts. You must define the following variables in the *pcbx_ug_model.ini* file.

Variable	Value options	Descriptions
`NxWriteHoleTypeAsCurves`	• `Pin` • `Via` • `Mounting` • `Tool` • `Other`	Imports holes as curves.
`NxWriteHoleTypeAsCutouts`		Imports holes as cutouts.

For each of the variables, you can define multiple values as a comma delimited list. These variables overwrite the `NxWriteHoleAs` variable.

Why should I use it?

You can represent certain types of holes in the board as curves or cutouts instead of filtering them out.

Where do I find it?

PCB Exchange looks for the *pcbx_ug_model.ini* file in the following locations:

- Network location specified by the `MAYA_PCB_ENV_DIR` environment variable.
- Working directory from where you launch PCB Exchange which is typically the same directory from which you launch NX.
- Location specified by the `MAYA_PCB_DIR` environment variable.
- The NX installation directory (*%UGII_BASE_DIR%\UGPCBXCHANGE*).

Creation of CAE items

What is it?

PCB Exchange now creates CAE items in Teamcenter so that you can use them to manage the information.

Vehicle Design

Pedestrian Protection enhancements

What is it?

When you perform the pedestrian protection analysis of a vehicle, you can now do the following.

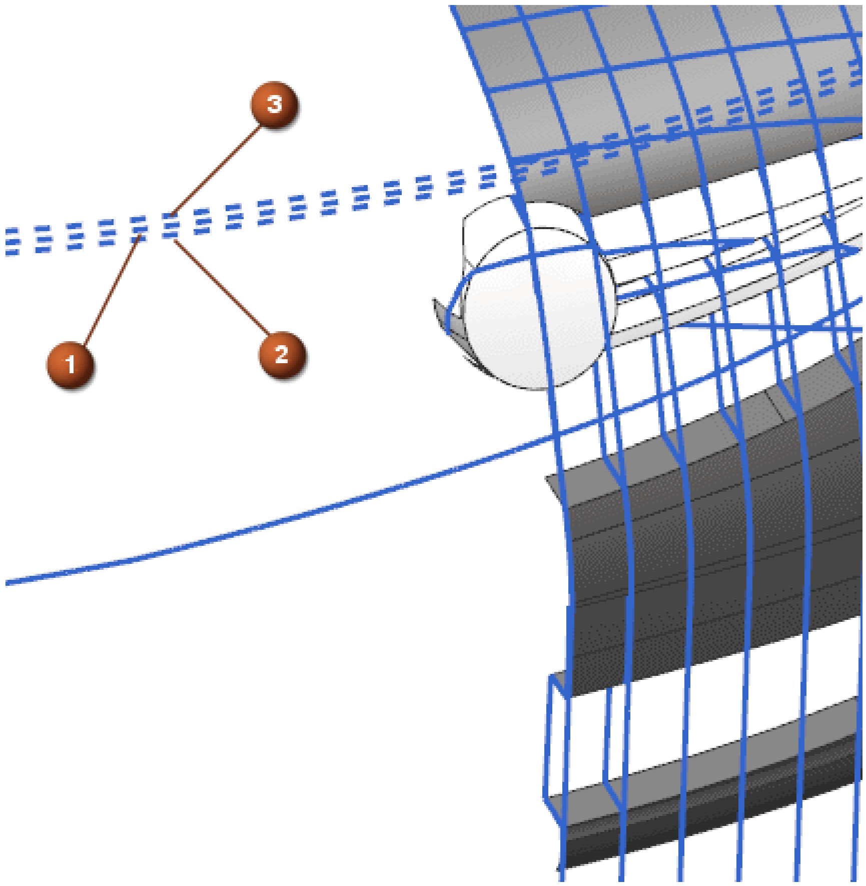
- Generate more accurate Bonnet Rear Reference Line (BRRL) output by controlling the rear reference step distance value.

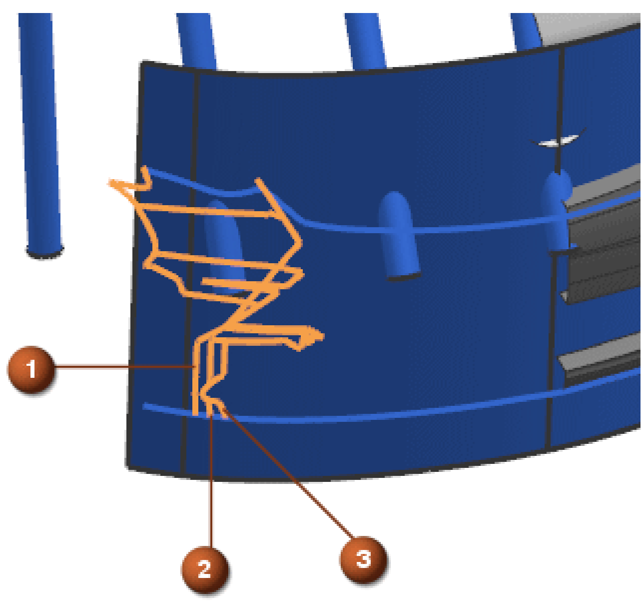
  Change the default value using the **Rear Reference Step Distance** customer default or enter a value in the **Rear Reference Step Distance** box in the **Sections** group in the **Standard Settings** dialog box.

  > **Tip**
  >
  > To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

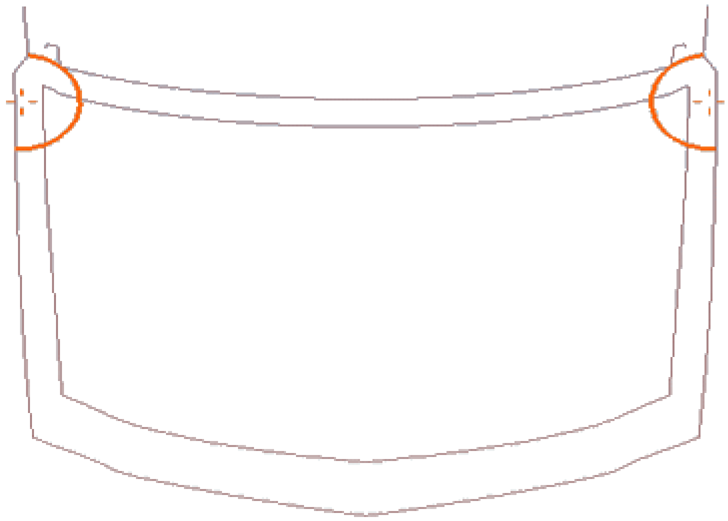
- Generate second marginal Bonnet Leading Edge (BLE) and second marginal Bumper Corner (BC) lines. These lines allow you to easily identify the worst case condition by comparing three sets of data.

  Change the default second marginal values using customer defaults or specify the values in the **Head Impact Reference Angles** group in the **Standard Settings** dialog box.





NX now generates circle templates at the corner of the BRRL and Side Reference Line (SRL), whether or not the lines touch. This lets you obtain a smooth curve at the corner of the BRRL and Side Reference Line (SRL) even if the lines touch.



NX now generates impact sheets using reference curves instead of impact curves. This lets you obtain a well-defined headform basin in the output.

Where do I find it?

Standard Settings dialog box

Application	Modeling
Prerequisite	You must define the vehicle coordinate system.
Command Finder	**Pedestrian Protection** 
Location in dialog box	**Pedestrian Protection** dialog box→**Edit Standards Settings** →**Standard Settings** dialog box

Second marginal values customer defaults

Command Finder	**Customer Defaults**
Location in dialog box	**Vehicle Design – General Packaging**→**Pedestrian Protection**→**Global Technical Regulation Standards** tab, **European Standards** tab, **Japanese Standards** tab, **Korean Standards** tab, or **Chinese Standards** tab

Vehicle Packaging option renamed

What is it?

The **Pedal Plane Angle (A47)** option name is changed to **Shoe Plane Angle (A47)**.

Where do I find it?

Standard Settings dialog box

Application	Modeling

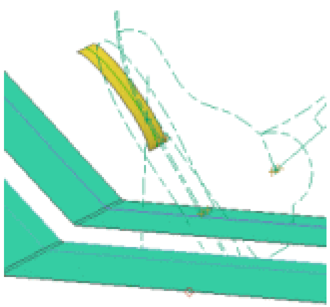
Prerequisite	You must define the vehicle coordinate system.
Command Finder	**Vehicle Packaging**
Location in dialog box	**Create Vehicle Packaging** wizard→**Occupant List** step→**Create** →**Create** wizard (for selected occupant)→**H-Point Design** step→**Position Method** group

H-point Design Tool enhancements

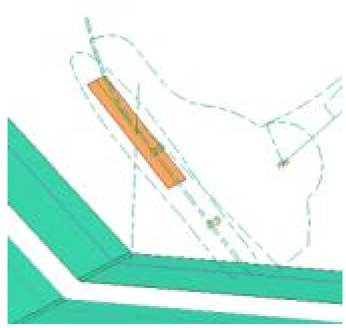
What is it?

You can now use the **H-point Design Tool** command to:

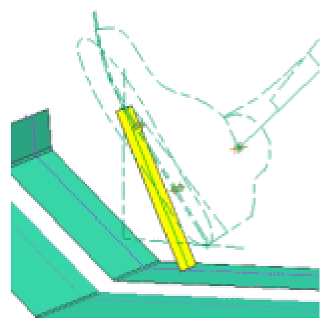
- Establish key vehicle reference points such as SgRP (Seating Reference Point) and Heal point in compliance with the SAE J4002 Feb2010 standards.
- Specify the pedal type as suspended curved, suspended flat, or treadle. To do this, use the **Select Pedal Type** drop-down list in the **Accelerator Shoe** step.



Suspended Curved Pedal

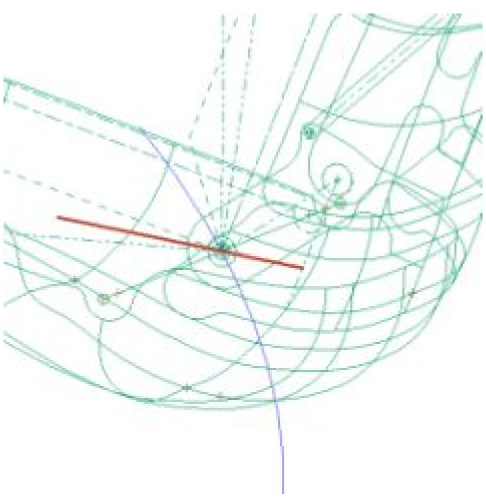


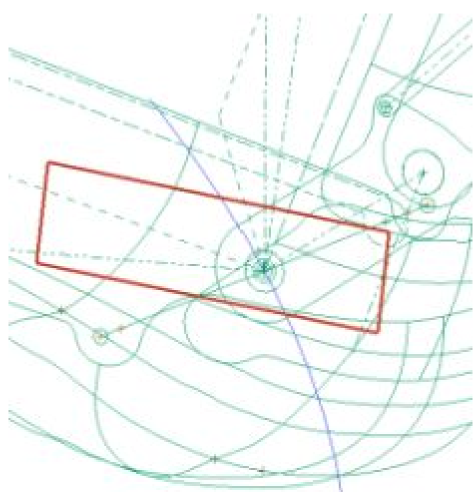
Suspended Flat Pivoting Pedal



Treadle Pedal

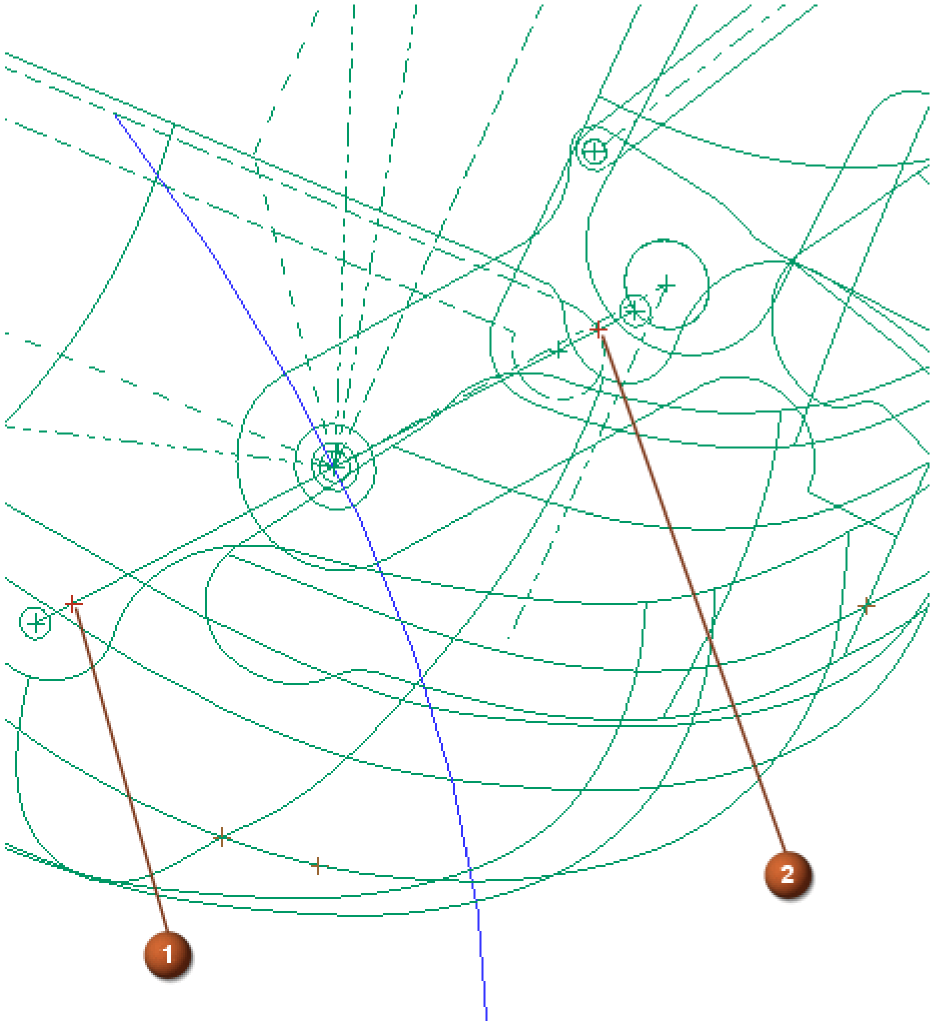
- Define and display 2–way or 4–way H-Point travel path.





2–way H-Point travel path 4–way H-Point travel path

- Get two new reference points (divot points) H2L and H2R in the output.



The following option names in the **H-Point Design Tool** wizard are changed.

Pre-NX 10 name	NX 10 name
Pedal Plane Angle (A47)	**Shoe Plane Angle (A47)**
By Pedal Geometry or Pedal Plane Angle	**By Pedal Geometry or Shoe Plane Angle**
PRP Width Coordinate (W1)	**BOFRP Width Coordinate (W1)**

The formula used to calculate the **Shoe Plane Angle (A47)** value is updated:

$SPA = 2.522(10^{-7})(H30^{3}) - 3.961(10^{-4})(H30^{2}) + 4.644(10^{-2})(H30) + 73.374$

Where do I find it?

Standard Settings dialog box

Application	Modeling
Prerequisite	You must define the vehicle coordinate system.
Command Finder	**H-point Design Tool** 

Chapter 4: CAM NX 9.0.1 — NX 10.0.1

General enhancements

Creating the IPW in the background

You can create the in-process workpiece (IPW) for drilling or milling operations in the background, while you work on other operations. To do this, you use the **Parallel Create 3D IPW** command. This command is similar to the **Parallel Generate** command that you use to generate operations in the background. Creating the IPW in the background is useful when:

- You do not want to wait for IPWs from earlier operations to be created before you simulate or generate an operation.
- You are going to use the **Show Thickness by Color** option to examine the IPWs for a series of operations.

When an IPW is being created for one or more operations:

- The symbol appears beside the nodes in the **Operation Navigator**.
- The **Stop Parallel Create 3D** option is available when you right-click an operation for which an IPW is being generated.
- The tool tip for the affected operations shows that the IPW is being generated and shows the operating system process ID of the background process.

After NX creates the IPW, the **IPW** column of the **Operation Navigator** shows the status of the IPW for each operation:

Generated

Not generated

Out of date

Where do I find it?

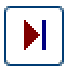
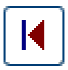
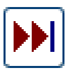
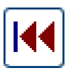
Initial release	NX 9.0.1
Application	Manufacturing
Prerequisite	Generate the operations before you attempt to create an IPW and then select them.
Ribbon bar	**Home tab**→**Workpiece** group→**Parallel Create 3D**

Operation Navigator	Right-click one or more selected operations→**Workpiece**→**Parallel Create 3D IPW**

Verification settings and playback enhancements

Keyboard shortcuts

When a playback button, for example **Step**, is selected, you can use keyboard keys to control the animation.

 **Step**	Right arrow key
 **Step Backward**	Left arrow key
 **Forward to Next Operation**	Page Up key
 **Rewind to Previous Operation**	Page Down key

For continuous motion, hold the corresponding arrow key.

Note

You can also use the arrow keys to move a selected slider or scroll a selected list.

Tool Path Visualization settings

The **Tool Path Visualization** dialog box settings are now retained throughout the NX session, with the following exception:

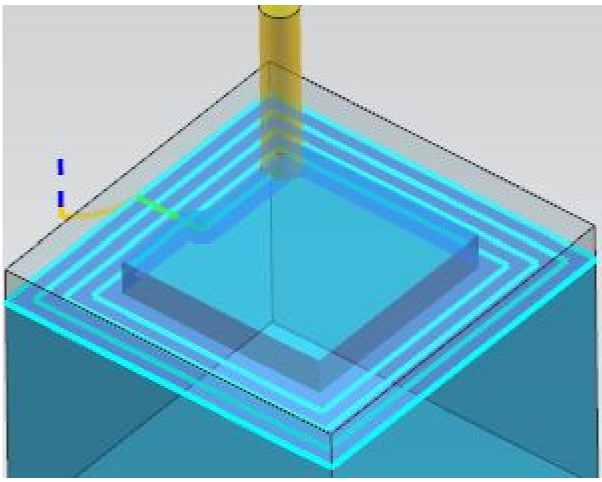
- The slider value that controls the listing in the path listing window is not retained.

Note

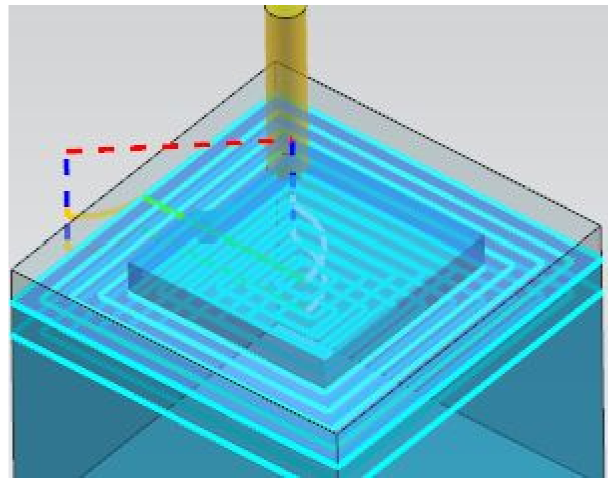
NX does not retain the settings in dialog boxes that you open from within the **Tool Path Visualization** dialog box.

Tool path display during playback

The **Tool Path** display options include a **Start to Current Motion** option, to display the path from the start point of the operation until the current position of the tool.



Start to Current Motion



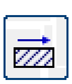
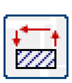
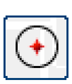
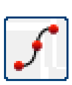
All

Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	You must have at least one operation that has been generated. For arrow key shortcuts, click a playback button to give it the focus. For settings retention, reopen the **Tool Path Verification** dialog box.
Location in dialog box	**Replay** or **3D Dynamic** tab→**Tool Path** list→**Start to Current Motion**

Displaying cutter paths - starting in NX 9.0.1

For milling and drilling operations, use the commands in the **Display** group of the **Home** tab to filter the display of tool paths for operations that you select in the **Operation Navigator**:

-  **Show Cutting Moves**
-  **Show Non Cutting Moves**
-  **Show Tool Center** This option affects ball tipped cutters only.
-  **End Point**

Note

- You no longer need to set the environment variable `UGII_CAM_TP_DISP_ENDPOINTS` to display end points.

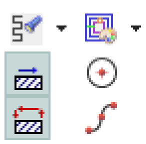
- If you turn off both **Show Cutting Moves** and **Show Non Cutting Moves**, NX turns off the path display for operations that you select in the **Operation Navigator**.

Where do I find it?

Initial release	NX 9.0.1
Application	Manufacturing
Prerequisite	Select an operation in the **Operation Navigator**
Ribbon bar	**Home** tab→**Display** group

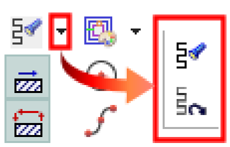
Displaying cutter paths - starting in NX 9.0.2

Display group

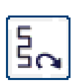


Use the two new options in the **Display** group of the **Home** tab to manage the display of tool paths when you select or edit an operation.

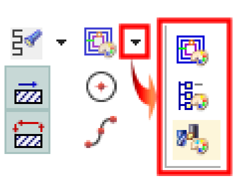
Display Drop-down



Use the **Display Tool Path**  option to show a tool path relatively quickly. The **Display** option supports the options in the display group.

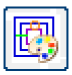
Use the **Replay Tool Path**  option to use all of the settings in the **Display Options** dialog box, including tool and path display options, replay speed, display of feed rates, arrows, and line numbers, and the generation options. You open the **Display Options** dialog box from the **Options** group when you are creating or editing an operation.

Coloring Drop-down



You can set the tool path display colors by motion type, by operation, or by tool.

The comparison images shown here are taken from one of the *sim06_mill_5ax_cam...* assemblies in your *...\MACH\samples\nc_simulation_samples* folder. The following three operations are selected in each image:

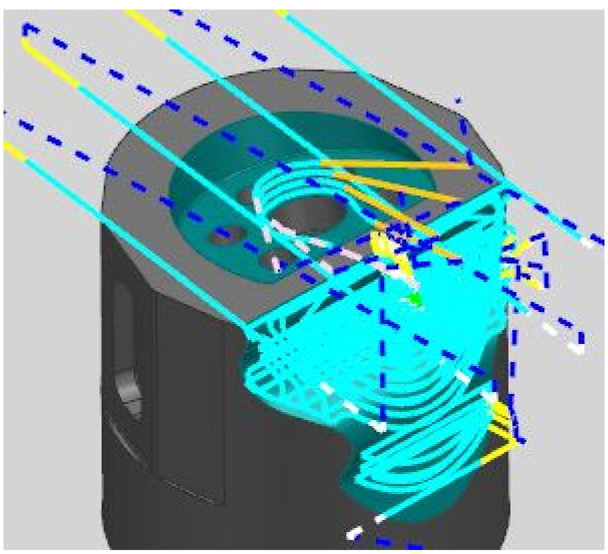


Motion Type

Use the **Motion type** option to retain the display behavior of versions previous to NX 9.0.2.

Tool path motions are displayed in the colors that you set in the **Options→Edit Display→Path Display Colors** dialog box. You can assign colors to different motions, such as engage, retract, rapid, and cutting moves.

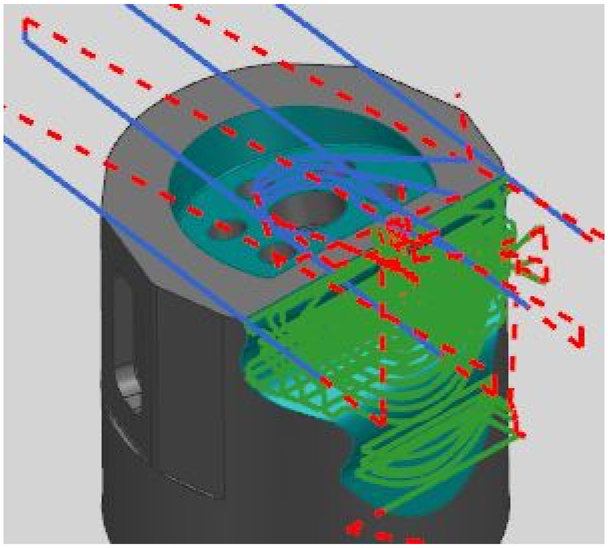
The example shows the default colors by motion type.



Tool

Use the **Tool** option to display the non-rapid moves of tool paths in a different color for each tool.

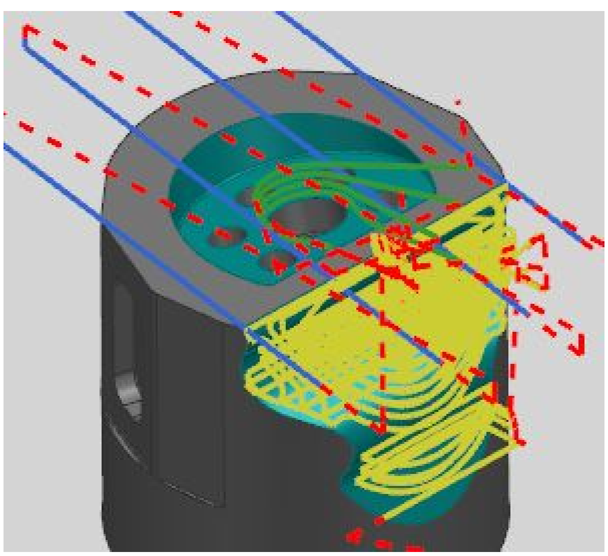
The example shows paths colored by tool. The first two paths are shown in a blue shade, and then there is a tool change. The third path, which uses a new tool, is shown in a green shade.



Operation

Use the **Operation** option to display the non-rapid moves of tool paths in a different color for each operation.

In the example, the path for the **FACE_TOP** operation is rendered in a blue shade, the path for the **CAVITY_TOP** operation is a green shade, and the path for the **CAVITY_MILL** operation is a yellow shade.



Customer defaults for tool and operation colors

The **Color by Tool or Operation** defaults control the default colors and order for tool path display by tool and by operation.

The **Rapid Motion**, **Color** and **Font** defaults control the font and color for rapid motions. Rapid motions are now red and dashed by default.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

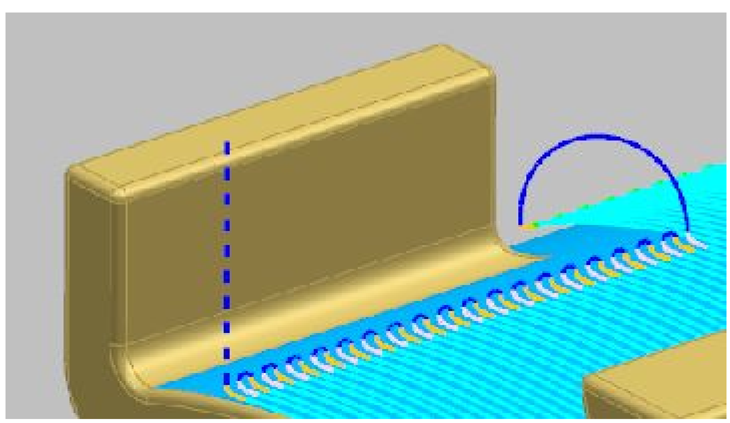
Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	Select operations in the **Operation Navigator** or edit an operation.
Ribbon bar	**Home** tab→**Display** group

Connecting tool path segments after edits

Use options in the **Transfer/Rapid** group of the **Trim Tool Path** dialog box to connect disjoint segments of tool paths that you have trimmed or divided with options in the **Tool Path Editor** dialog box.

Transfer at	**Direct**	Makes a direct connection between the end of the retract of a cutting motion to the start of the engage of the next cutting motion.
	Clearance	Sets the clearance height for the transfer.
	Lowest Safe Z	Applies a direct move first. If the move is not gouge free, NX uses a previous safe Z-level plane.
	Smooth	Creates a default transfer with no selectable options.



Smooth connections

	Plane	Lets you define a plane for transfer moves for each trim that you perform.
Transfer using	**None**	Does not use any specific transfer motions.
	Ramp on Shape	Creates a ramping engage move that follows the shape of the first cutting motions.
	Lift and Plunge	Produces engages and retracts with vertical moves. Enter a **Lift and Plunge** Height.
Height	Sets the height at which the transfer takes place.	

Where do I find it?

Initial release	NX 9.0.1
Application	Manufacturing
Command Finder	Edit tool path
Operation Navigator	Right-click operation→**Tool Path**→**Edit**
Location in dialog box	[**Tool Path Editor**] dialog box→**Edit Actions** group→**Trim** →[**Trim Tool Paths**] dialog box→**Transfer/Rapid** group

Trimming tool paths in the Tool Path Editor

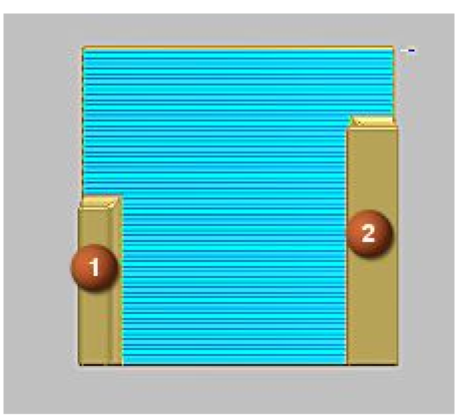
You can trim a tool path by selecting an area in the current graphics view plane. The selection tools include:

- Rectangle
- Lasso
- Polygon

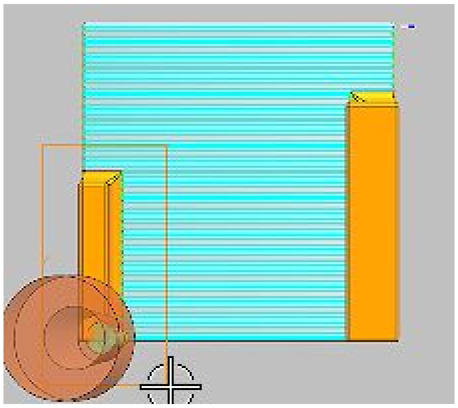
Tip

To add the polygon **Select by Polygon** method, use the **Command Finder** to find **Select by Polygon** . Right-click and choose **Add to Top Border Bar**. You can further customize the Top Border bar by dragging the icon into the **Selection** list.

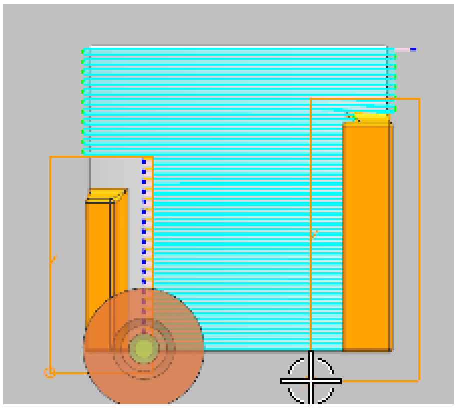
You can specify smooth connections between the truncated cutting motions. In the tool path shown, the walls marked (1) and (2) are colliding with the tool holder. You can trim those areas from the tool path.



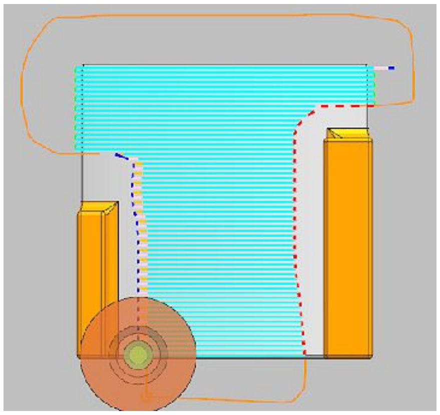
You can remove one area by trimming the motions that are inside a rectangle, as shown.



To complete the edit, repeat the trim using a second inside-rectangle trim.

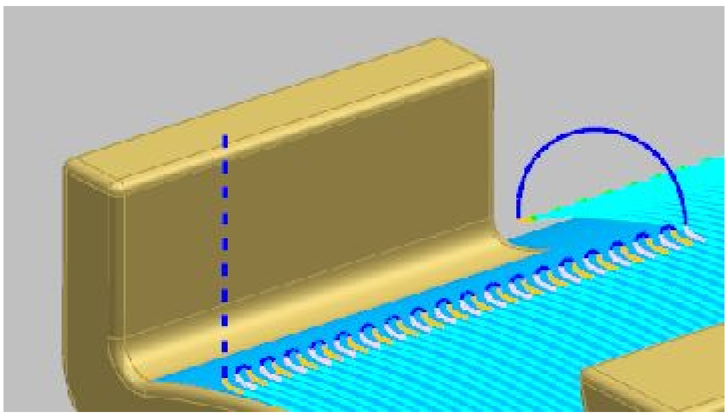


You can achieve the same result in a single trim operation by trimming outside a selected area using the **Lasso** (shown) or **Polygon** selection method.

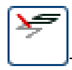


Smooth connections between trimmed motions

The following figure shows the trimmed path, with the smooth connections option.



Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	You must edit a generated tool path.
Command Finder	Edit Tool Path
Operation Navigator	[Select operation]→right-click→**Tool Path**→**Edit**
Location in dialog box	**Edit Actions**→**Trim** →**Trim Tool Paths** dialog box→**Trim Geometry** group→**Geometry** = **Selection in View** Choose the trim method on the Top Border bar.

IPW rendering improvements (10.0.1)

What is it?

Needle count defaults for rendering the IPW are higher counts than in earlier releases. This is now practical due to increased performance in NX and much more capable computers than were available when the defaults were created.

Needle Count Option	Previous versions	NX 9.0.3, NX 10.0.1 and later
Coarse	200	350
Medium	350	500
Fine	500	1000

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Menu	**Menu**→**Preferences**→**Manufacturing**
Location in dialog box	**Geometry** tab→**In Process Workpiece Tolerances**

IPW tolerances moved into the part (10.0.1)

What is it?

When you have a part that requires IPW settings that are different from those set by the **IPW Tolerances** group in the customer defaults, **Chordal Tolerance**, **Tolerance Mode**, **Needle Distance**, and **Needle Count**, you can now override the defaults in the **Manufacturing Preferences** dialog box. These settings are saved with your part.

The customer defaults now serve only to establish default tolerance values for the **Manufacturing Preferences** dialog box.

Why should I use it?

Part-specific preferences are useful when you have parts that require special settings, for example a few large parts that require a higher needle count than your other parts.

You can change the IPW tolerances and see the effect without having to restart your NX session.

You can create template parts that have specific settings for the various types of parts with which you deal regularly.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Menu	**Menu→Preferences→Manufacturing**
Location in dialog box	**Geometry** tab→**In Process Workpiece Tolerances**

Generating the IPW with the operation (10.0.1)

What is it?

A new customer default, **Generate IPW with Path**, lets you generate the in-process workpiece (IPW) as you generate an operation. NX saves the IPW with the operation.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

Why should I use it?

Saving the IPW with the operation saves time when you do the following:

- Use the **Verify Tool Path**→**3D Dynamic** method.
- Select the **Show Thickness by Color** option or the **Workpiece**→**Show 3D** option.
- Generate operations that use the previous IPW.

You can:

- Visually inspect the operation result as soon as it is generated.
- Immediately verify the tool path to check for collisions with the IPW.

Where do I find it?

Initial release	NX 9.0.3 MP2
Application	Manufacturing
Customer Defaults	Search for **Generate IPW with Path**

Toolpath verification and analysis consolidation (10.0.1)

What is it?

In previous releases, new functionality was added beside existing functionality. This resulted in some duplication and confusion over which command to use for a given task. Some older functions are now hidden by default in the **Tool Path Verification** dialog box.

2D Dynamic tab

The **2D Dynamic** tab is now hidden. This protects you from issues that might arise if, for example, a collision occurs but is not visible in the selected 2D view.

To restore the **2D Dynamic** tab, select the **Show 2D Dynamic Page** customer default.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

3D Dynamic tab

To ensure maximum reliability, on the **3D Dynamic** tab, the **Facetted Solid** options are hidden.

We recommend that you click **Show Thickness by Color** to check the amount of excess material that remains on the part.

We recommend that you use the following methods for gouge checking:

- Right-click an operation in the **Operation Navigator** and choose **Tool Path**→**Gouge Check**.
- In the **Tool Path Verification** dialog box, on the **Replay** tab, click **Gouge and Collision Settings**.

To restore the **Facetted Solid** options, select the **Enable Facetted Solid for Gouges and Excess** customer default.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing

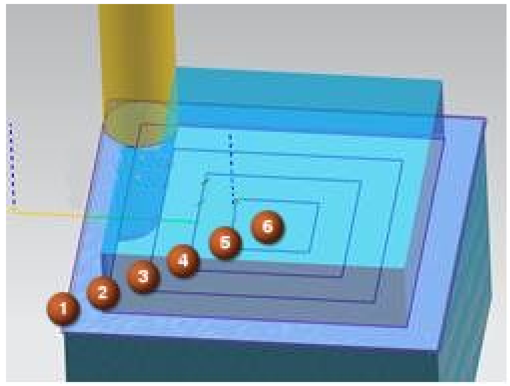
Milling

Automatic cut pattern direction

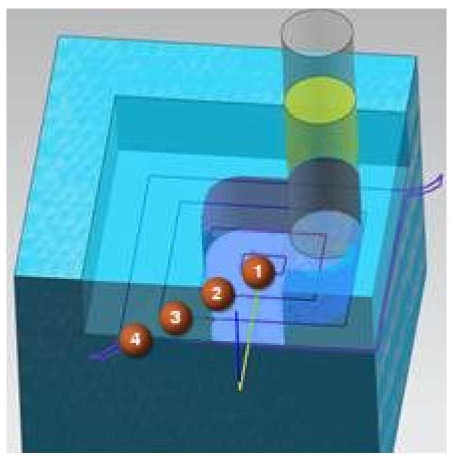
What is it?

You can choose the **Automatic** pattern direction option in a **Follow Periphery** cut pattern. The pattern direction will alternate between **Inward** and **Outward** per cut level and cut region condition. NX does the following.

- With an open periphery, the cutter approaches from outside the part and engages the material while working towards the center of the part.

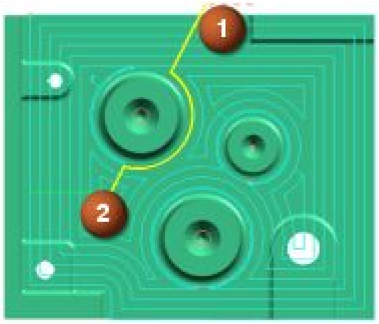


- With a partial or completely closed periphery, the cutter approaches from outside the part and the first initial entry machines a slot to reach the region start point. From there the toolpath continues to work its way outward.

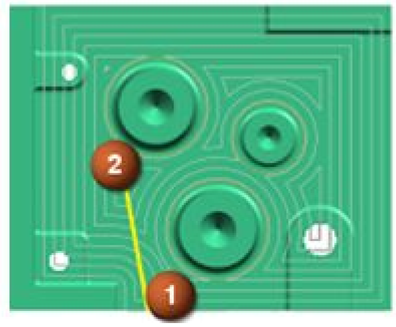


- For an **Outward** pattern with standing bosses or features, the entry point and path to the region start point are optimized.

Previous NX results



NX 9.0.2 results

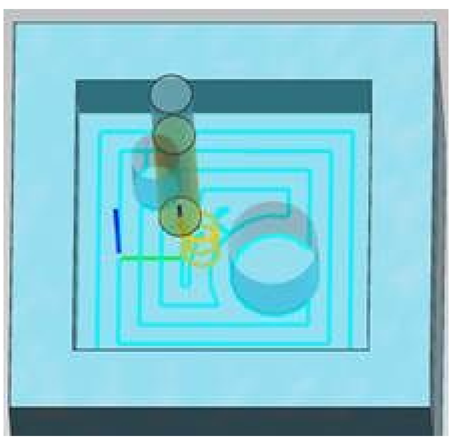


1 = Entry point

2 = Region start point

- Pockets, voids, or existing holes are considered for the open engage portion of the tool path.

Previous NX results



NX 9.0.2 results

Why should I use it?

By using the **Automatic** pattern direction option, you allow NX to optimize the created cavity milling tool paths for maximum efficiency, tool life and performance.

Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Location in dialog box	**Cutting Parameters** dialog box→**Strategy** tab→**Cutting** group→**Pattern Direction**→**Automatic**

Cavity Mill Cutting Parameters (10.0.1)

What is it?

When you create a tool path that uses the **Follow Periphery** cut pattern, the **Legacy** options in the **More** tab in the **Cutting Parameters** dialog box are no longer available when you set the **Pattern Direction** to **Automatic**.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Location in dialog box	**Cutting Parameters** dialog box→**Strategy** tab→**Pattern Direction** list→ **Automatic**

Improving surface finish on corners and fillets Curve/Point drive method

When you use the **Curve/Point** drive method, you can do the following:

- Position the tool to follow along edges of the part geometry at a specified offset using the **Offset Left** option.
- Shift the tool contact point a distance along the curve tangent of a non-center cutting tool such as a bull nose end mill. When you cut around an axis, this is equivalent to a rotational shift around the axis.
- Chamfer.

These enhancements are primarily intended to improve the surface finish for 5-axis machining of rotary floors. The **Curve/Point** drive method is not used exclusively for rotary floors, and the enhancements may be useful for other types of applications. When the tool axis is not perpendicular to the cut area, you may see differences in the 3-axis tool path for the **Offset Left** option. See the *Manufacturing product notes* section in the *Release Notes* for details.

Following edges at an offset

NX does the following to create the tool path:

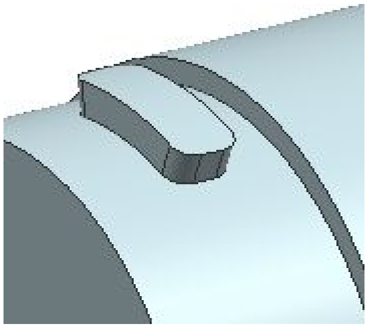
1. Offsets the selected edges or curves along the cut area surface to create the drive curves.
2. Checks for gouges against part or check geometry, and removes any portions of the tool path that gouge. If the offset value is too small to clear the geometry, NX does not create a tool path.

   Note

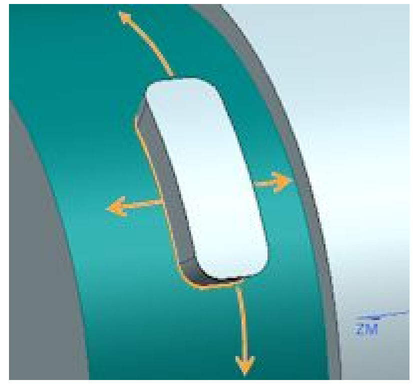
   To specify an offset to the right, enter a negative value.

In the following example, the drive curves are selected from the feature edges.

Part to machine



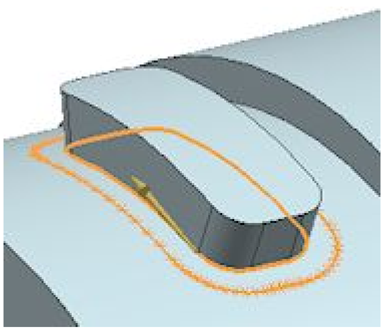
Cut area and edges to offset



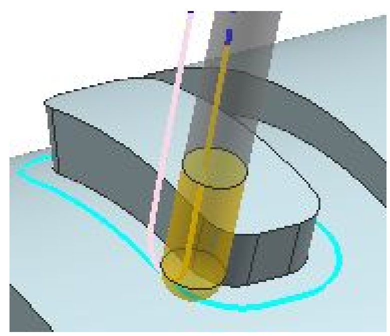
Tool diameter = 3 mm

Offset Left = 1.5 mm

Drive curves



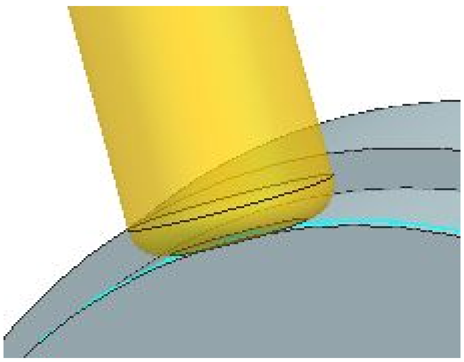
Tool path



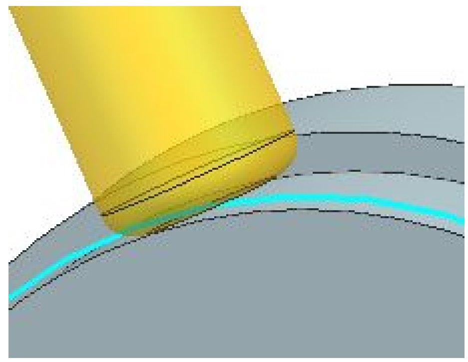
Shifting the contact

The following example shows the effect of shifting the tool contact by 1.5 mm on a 3 mm diameter tool.

Tool at default contact point

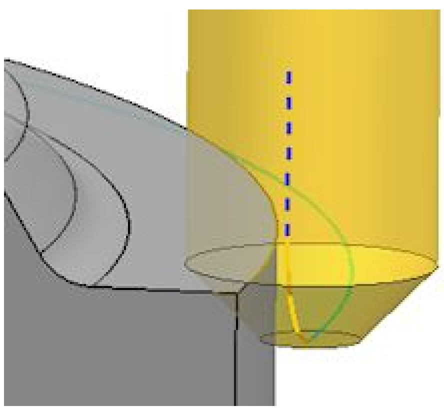


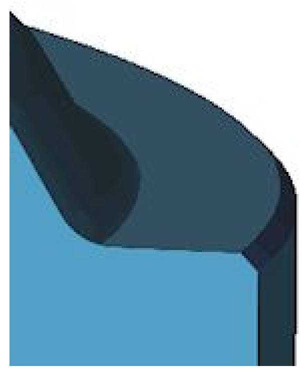
Tool contact shifted



Chamfering with Curve/Point

Use a chamfering tool and specify negative stock to cut a chamfer. When you generate the operation NX displays a warning message. To continue generating the operation, click **No** in the message box.





Why should I use it?

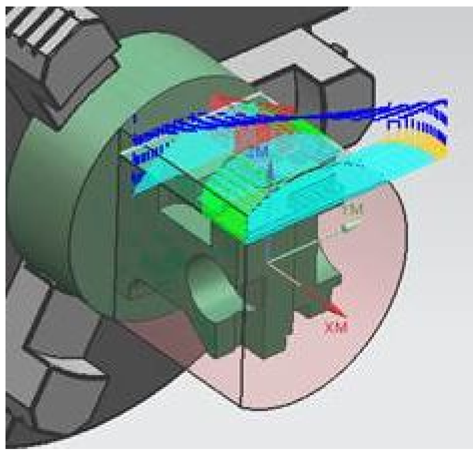
These enhancements help improve surface finish quality. Previously, there was no way to shift the tool contact, and it was necessary to create offset curves. You no longer need to create offset curves to drive the tool along part features.

Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	**Curve/Point** drive method
Location in dialog box	**Curve/Point Drive Method** dialog box→**Drive Settings** group→**Offset Left** and **Tool Contact Shift**

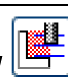
Performance enhancements for Floor Wall operations using a 3D IPW

What is it?



The time required to generate tool paths, or to edit an operation with the **Preview** option selected is now improved. The performance improvements depend on the complexity of the blank and 3D IPW, and are most noticeable when the operation uses a complex or nonrectangular blank.

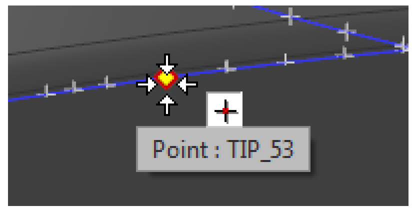
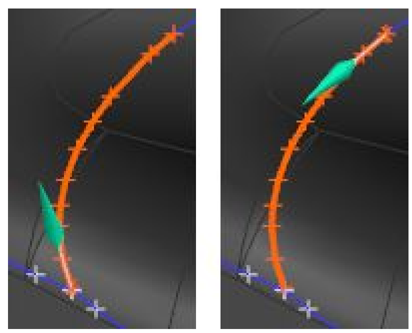
Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Command Finder	**Floor and Wall with IPW** 

Cut regions and cut order in Flowcut Single operations

The **Cut Regions** dialog box appears when you generate a **Flowcut Single** operation, like the **Manual Assembly** dialog box in NX 9.0 and earlier versions. Segments from earlier versions are now referenced as regions.

Use the **Flowcut Single** operation **Cut Regions** dialog box to reorder, divide, or delete regions of a flowcut path.

Regions to Cut group	**Name** column	Shows the cut regions in the current cut order. To reorder the list, drag a row to a new position. You can also rename a region in this column.
	 **Divide**	Lets you specify a point to divide the selected region. NX uses the nearest point on the region to the place where you click as the dividing point. The region points are highlighted when the dialog box is open. Point numbers appear when the mouse cursor is over a point. 
	 **Reverse**	Reverses the cut direction of the selected region. NX displays a direction arrow at the start point of the region. 
	 **Delete**	Removes the selected region from the tool path.

Where do I find it?

Initial release	NX 9.0.1
Application	Manufacturing
Prerequisite	Create or edit a **Flowcut Single** operation, and generate the operation

Cut region control for Flowcut Reference Tool

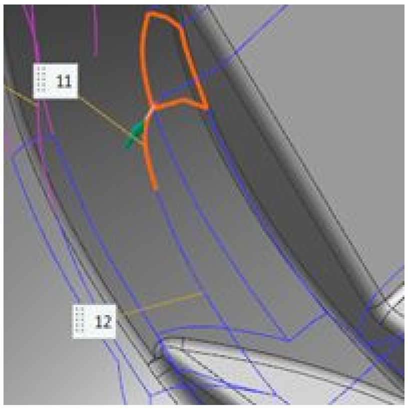
What is it?

You can subdivide the cut area for a **Flowcut Reference Tool** operation. To do this, set the **Cut Order** list to **User Defined** in the **Flow Cut Drive Method** dialog box. NX initially subdivides the cut area based on the steepness angle you provide.

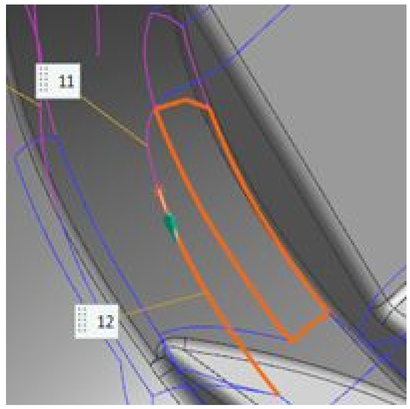
Region sizes

You control the region sizes. Merge or divide the regions as required. NX updates the cut region display as you make modifications.

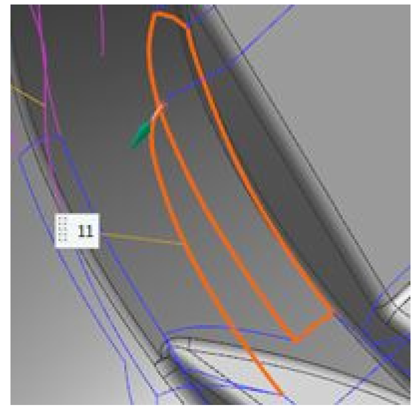
Example of merged regions



Region 11



Region 12



Regions merged as one

Region names when created:

<operation_name> _R_1

<operation_name> _R_2

When you divide regions, NX will append the region name with a sequential numerical value. For example, if you divide **Operation_R_1**, the resulting region names would be:

Operation_R_1_1

Operation_R_1_2

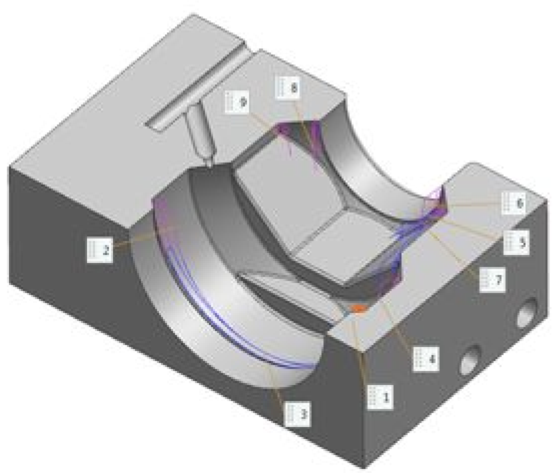
Cut regions and cutting sequence

You control the regions to cut and their cutting sequence. Delete regions to completely avoid cutting certain areas.

Cut Regions list example:

Name	Cut Order	Containment	Type
Operation Regions			
FLOWCUT_REF_TOOL_R_1	1		Flow
FLOWCUT_REF_TOOL_R_1	2		Flow
FLOWCUT_REF_TOOL_R_1	3		Flow
FLOWCUT_REF_TOOL_R_1	4		Flow
FLOWCUT_REF_TOOL_R_1	5		Flow
FLOWCUT_REF_TOOL_R_1	6		Flow
FLOWCUT_REF_TOOL_R_1	7		Flow
FLOWCUT_REF_TOOL_R_1	8		Flow
FLOWCUT_REF_TOOL_R_1	9		Flow

Cut order labels in the graphics display correlate to the **Cut Regions** list.



You can also do the following:

- Control the cut pattern and other tool path parameters within each cut region. Define a region as steep or non-steep, and NX applies the appropriate Flowcut Drive Method settings.
- Change the containment type for the region.
- Reverse the flowcut region cut direction. NX will not reverse an area region.

Editing updates

You can now do the following:

- Drag regions to change the cut order.
- Undo changes from the current editing session.
- Save the current cutting sequence. NX saves the edits with the operation each time it generates the operation, and uses the saved sequence to produce the updated tool paths.

Why should I use it?

Managing regions can improve cutting efficiency, improve the part surface finish, and improve cutter wear and performance.

Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	A **Flowcut Reference Tool** operation
Location in dialog box	**Flowcut Ref Tool** dialog box→**Drive Method** group→**Method** row→**Edit** **Flow Cut Drive Method** dialog box→**Output** group→**Cut Order**→**User defined**→**OK** **Flowcut Ref Tool** dialog box→**Actions** group→**Generate** →**Cut Regions** dialog box

Flowcut Region Management (10.0.1)

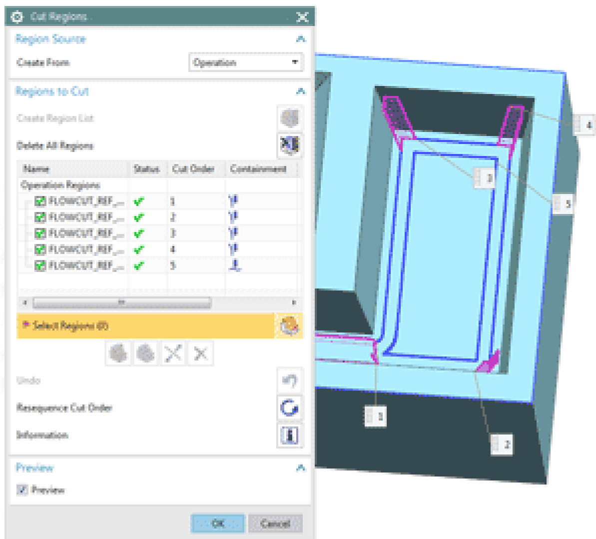
What is it?

In a Flowcut operation, you can create, visualize, edit, and save cut regions without generating the tool path. **Cut Region** options will be available in the **Flow Cut** geometry group.

You Can:

- Generate the tool path based on the saved cut regions within the operation.
- View the regions graphically.
- Import inactive regions from previous sessions into the current session.

When changes are made to the parameters of a region, NX displays a message and marks the region with an Invalid icon in the region list until the operation is regenerated.



Why should I use it?

Because you can edit cut regions before you generate the operation, overall processing time can be reduced.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Flowcut Operation
Location in dialog box	**Flow Cut Operation** dialog box

Corner smoothing for Z-level non-steep cut patterns (10.0.1)

What is it?

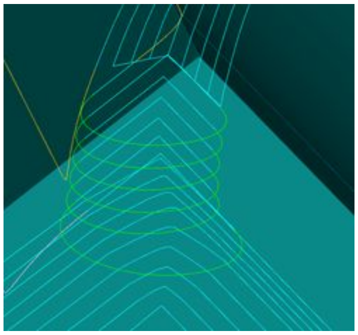
You can smooth corners in **Flow Cut** operations with **Zig**, **ZigZag**, or **ZigZag with Lifts** patterns for non-steep regions. To do this, you:

1. Set steep regions to **Zlevel**.

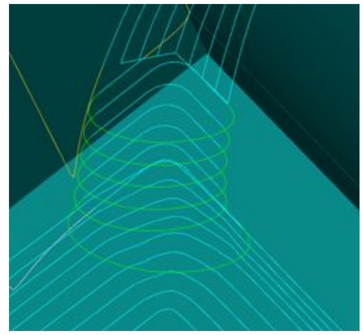
2. Set the **Smoothing** cutting parameter to **All Passes**.

3. Specify a radius size to be applied to the corners.

4. Generate the operation.



☐ **Corner Smoothing**



☑ **Corner Smoothing**

Why should I use it?

When you machine hard material, or machine at high speeds, consider adding fillets to all corners. Corner fillets can do the following:

- Prevent a sudden change in direction and cutter deflection which can cause excessive stress on the machine tool and cutter.

- Aid in tool path generation for Nurbs output because smooth transitions are easier to blend into Nurbs than sharp corners.

The **Smoothing** option gives you better control of tool path output for finishing operations and improves machining efficiency.

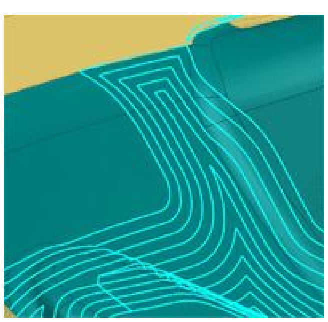
Where do I find it?

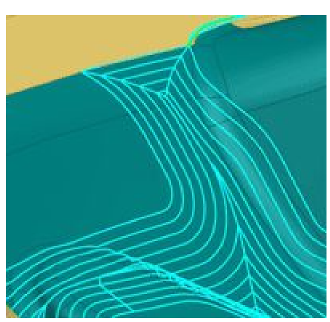
Initial release	NX 9.0.3 MP2
Application	Manufacturing
Location in dialog box	**Flowcut Reference Tool** operation→**Cutting Parameters**→**Corners tab**→**Path Shape in Corners** group→**Smoothing**→**All Passes**

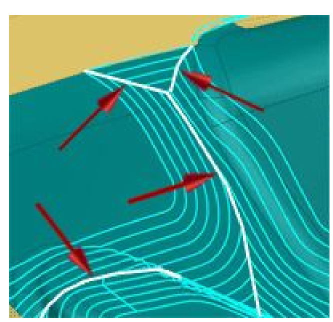
Adding center passes in Flowcut operations (10.0.1)

What is it?

When you machine corners in Flow Cut operations, you can reduce or remove scallops. To do this, use the **Add Center Passes** option. This option creates a center pass in the region when space between offset passes produces a larger than desired gap that results in a scallop.







☐ Add Center Passes ☑ Add Center Passes Added paths highlighted.

Why should I use it?

This option helps you to improve surface finish when you machine corners.

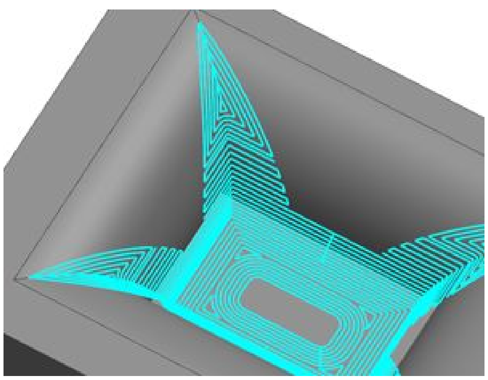
Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Flowcut Operation
Location in dialog box	**Flow Cut Drive Method** dialog box→**Non-steep Cutting** and **Steep Cutting** groups

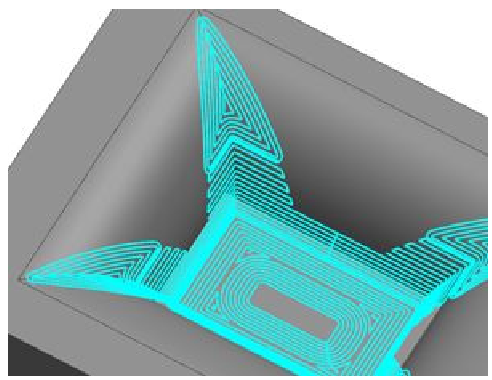
Using Steep Overlap and Follow Periphery in Flowcut operations (10.0.1)

What is it?

You can overlap the cutting motions between steep and non-steep regions in Flow Cut operations. Use the **Steep Overlap** option to specify the distance or a percentage of the tool diameter overlap. Additionally, you can use a Follow Periphery pattern for both steep and nonsteep regions.



No **Steep Overlap**



Steep Overlap set at 0.100”

Why should I use it?

When you overlap the cut motions, you can sometimes achieve a smoother finish.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	**Flowcut** operation using a **Zlevel** steep pattern with a **Follow Periphery** non-steep pattern.
Location in dialog box	**Flow Cut Drive Method** dialog box→**Steep Cutting** group

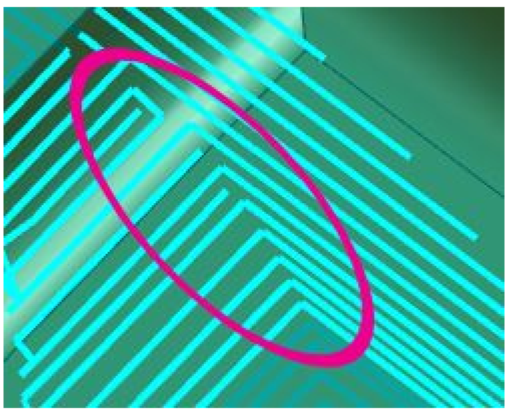
Corner smoothing

What is it?

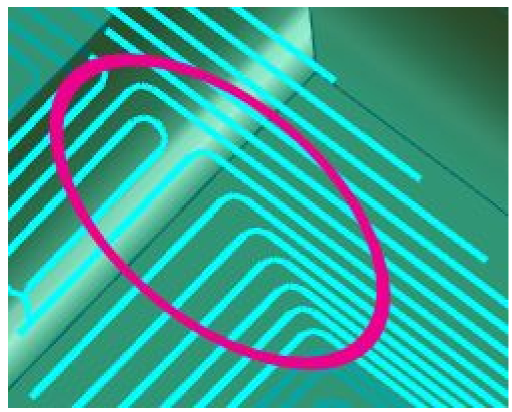
You can smooth corners in Zlevel cut patterns in **Flowcut Reference Tool** operations. To do this, you:

- Set steep and non steep regions to **Zlevel**.
- Select the **Corner Smoothing** cutting parameter.
- Specify a **Radius** size to be applied to the corners.

Previous versions of NX



NX 9.0.2



Why should I use it?

When you machine hard material, or machine at high speeds, consider adding fillets to all corners. Corner fillets can do the following:

- Prevent a sudden change in direction and cutter deflection which can cause excessive stress on the machine tool and cutter.
- Aid in tool path generation for Nurbs output because smooth transitions are easier to blend into Nurbs than sharp corners.

The **Corner Smoothing** option gives you better control of tool path output for finishing operations and improves machining efficiency.

Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Location in dialog box	[**Flowcut Reference Tool** operation]→**Cutting Parameters**→**Corners** tab→**Path Shape in Corners** group→**Smoothing** row→**All Passes**

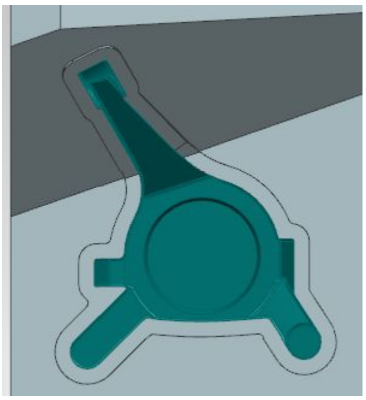
Using a boundary with the Area Milling drive method

Specify Trim Boundaries

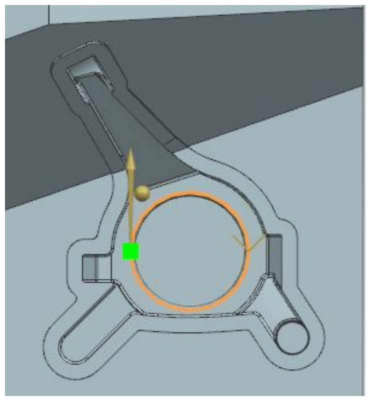
Use the **Specify Trim Boundaries** command to control cut areas in contouring operations that use the **Area Milling** drive method.

Contain the cut area by one or more loops of bounding objects. Boundaries can be edges, any type of curves including sketch curves, or points. Boundaries are projected onto the part surfaces along the normal vector of the boundary plane.

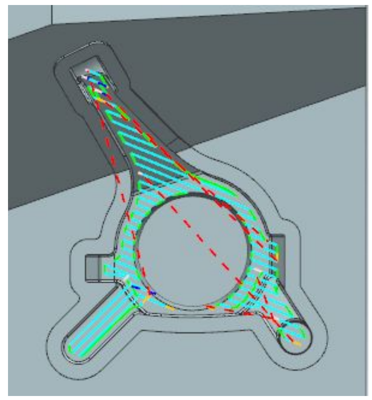
Edge as boundary



Cut Region

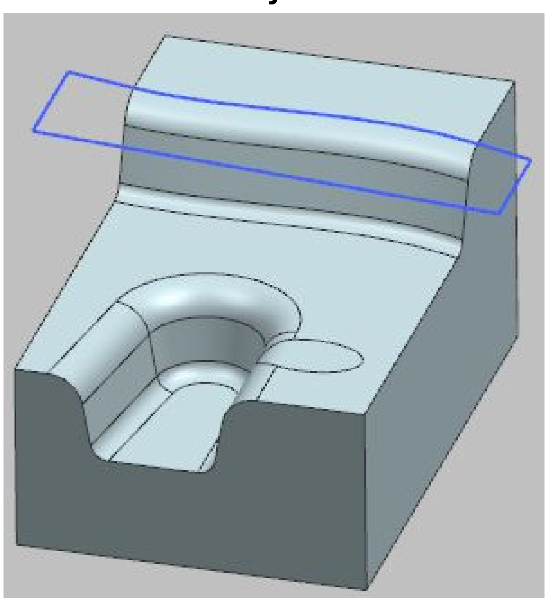


Boundary edge, trim inside

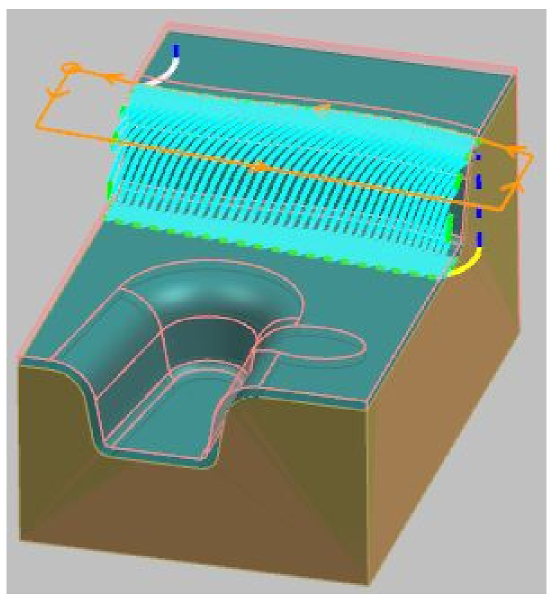


Tool Path

Sketch as boundary



Boundary sketch, trim outside

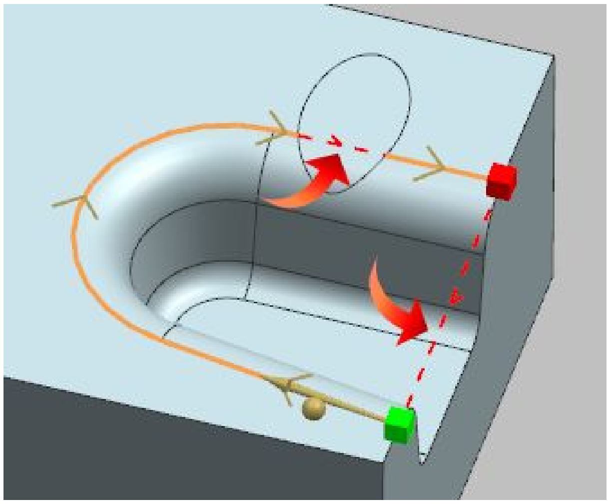


Tool Path

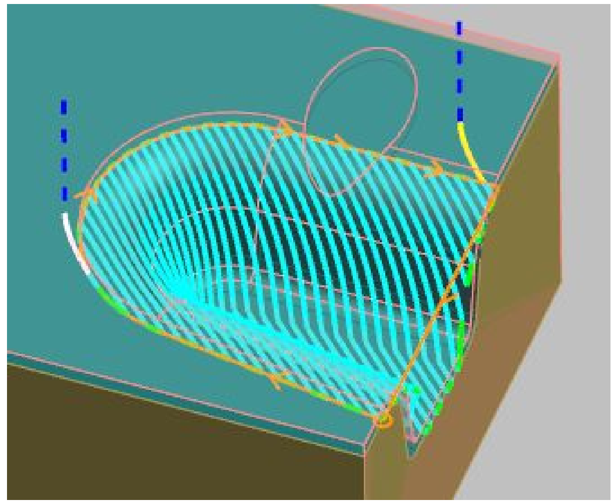
Closing gaps

You can leave gaps between curves or edges. The gaps are automatically closed by straight lines.

Closing gaps



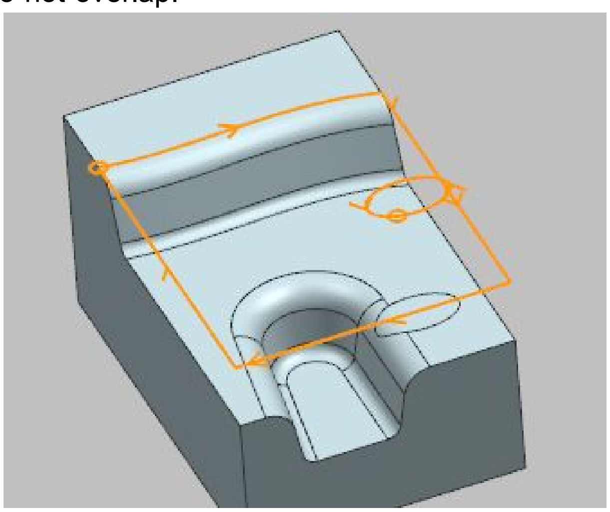
The arrows indicate closed gaps, boundary trim outside



Tool Path

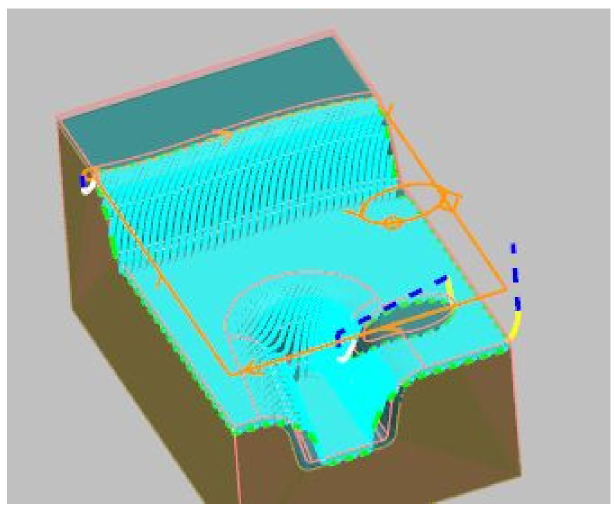
Nested boundaries, no overlap

You can define nested containment boundaries provided they indicate a clear trimming area and do not overlap.



Outer boundary trim outside

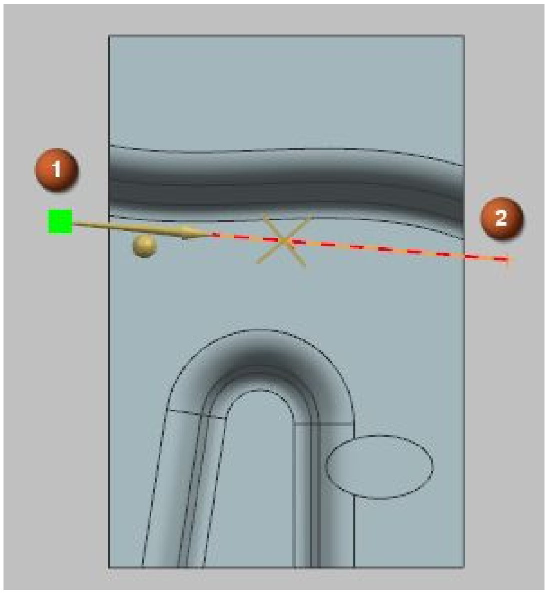
Inner boundary, trim inside



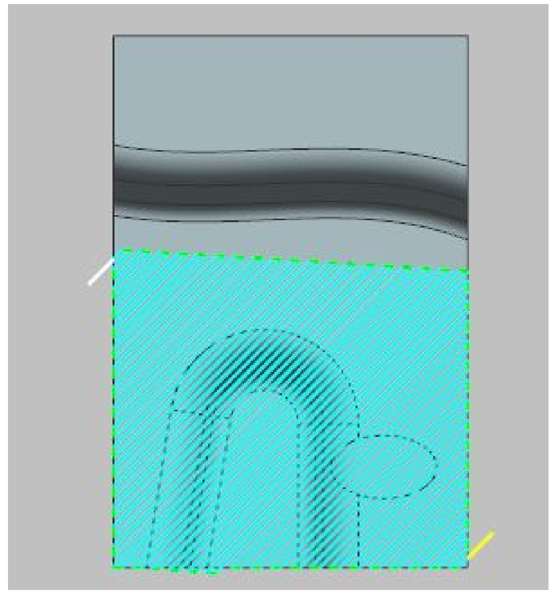
Tool Path

Linear open boundary

You can define open boundaries provided that the open ends project off the part.



Mouse clicks at (1) and (2) define two points, off part, as an open boundary



Tool Path

Trim boundary options

The trim boundary options are always available in the **Contour Area** operation subtype. They appear in the following operation subtypes after you select the **Area Milling** drive method :

- **Fixed Contour**
- **Contour Surface Area**

Contact tool position

When you specify the **Contact** tool position, NX moves the tool to contact the projected boundary in such a way that the entire area is cut.

Area milling geometry with boundaries

In the **Geometry** view of the **Operation Navigator**, you can create a **MILL_AREA** geometry group and include trim boundaries in the definition. You can then reuse the geometry area in multiple operations.

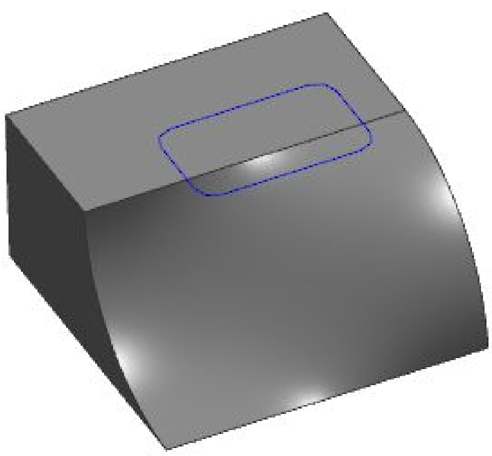
NX displays this type of geometry group in the initial **Geometry** list only when you choose the **Contour Area** operation subtype. If you create **Fixed Contour** or **Contour Surface Area** operation subtypes, you must edit the drive method to **Area Milling** before the area mill geometry appears in the **Geometry** list.

Where do I find it?

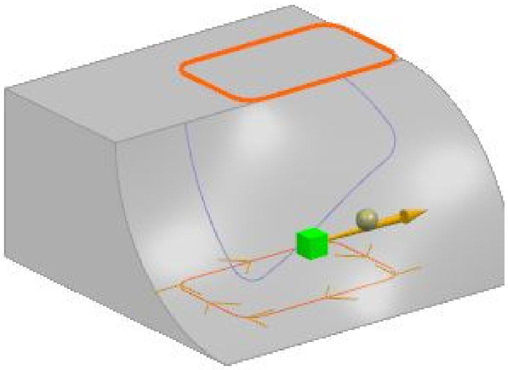
Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	You must have a contouring operation that uses the **Area Milling** drive method.
Location in dialog box	**Geometry** group→**Specify Trim Boundaries** 

Contact tool position on Area Milling boundaries

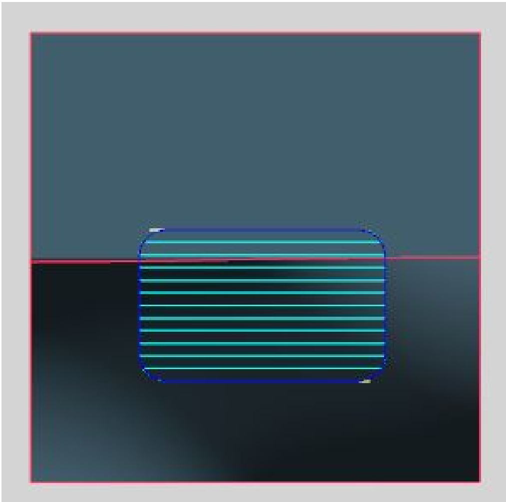
When you specify the **Contact** tool position, NX moves the tool to contact the projected boundary in such a way that the entire area is cut.



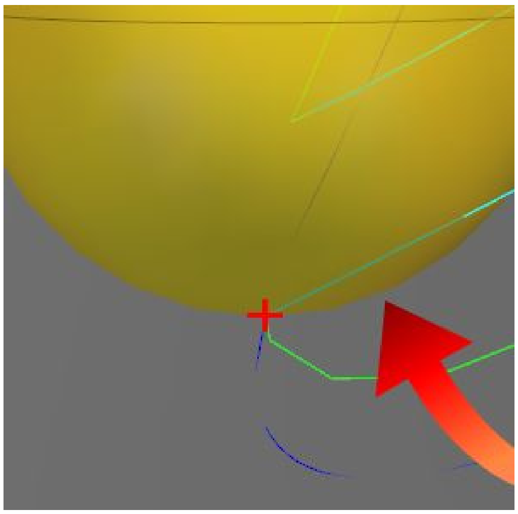
The part for this example has a surface that has both concave and convex regions. A sketched rounded rectangle is the boundary.



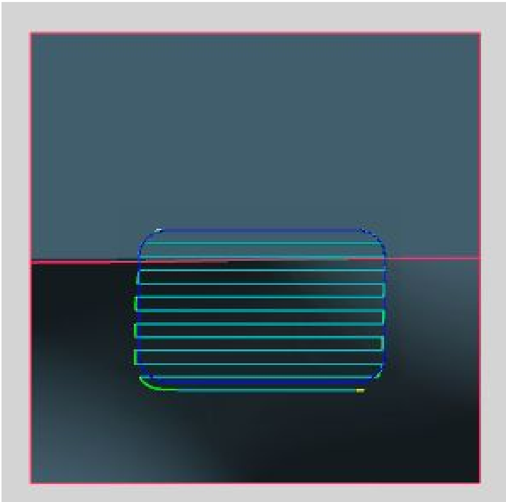
When projected along the tool axis, the Z-axis in this example, the boundary encompasses both concave and convex surface areas.



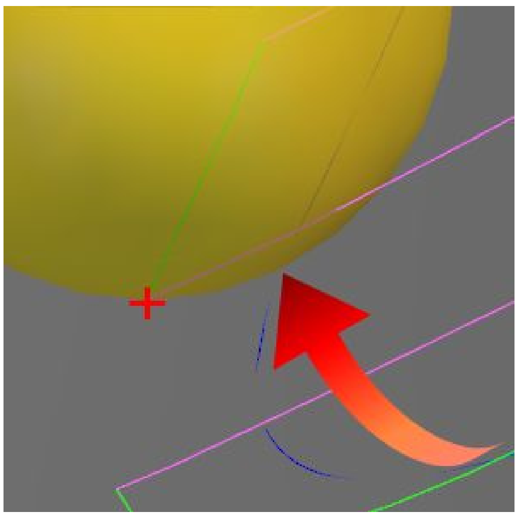
As seen along the tool axis, a tool path with the **On** tool position moves the tool tip strictly within the boundary.



When you use the **On** tool position, in steep areas, small concave areas are not completely cut, and small convex areas are cut outside of the boundary. This happens because when the tool tip is positioned over the boundary, the point of contact of the tool with the face is some distance from the boundary.



As seen along the tool axis, a path with the **Contact** tool position moves the tool tip beyond the boundary, or stops short of the boundary, depending on the nature of the surface slope.



The **Contact** option positions the tool tip such that the point of contact of the tool with the face lies on the boundary. This ensures that the tool contacts all points that lie inside the boundary, and does not contact points that lie outside of the boundary.

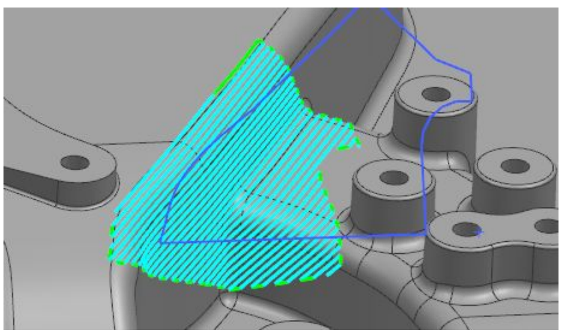
Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	Create or edit an operation that uses the **Area Milling** drive method, and create or edit boundaries to contain the tool motion.
Operation dialog box	**Specify Trim Boundaries** 
Location in dialog box	**Trim Boundaries** dialog box→**Boundaries** group→**Tool Position** list→**Contact**

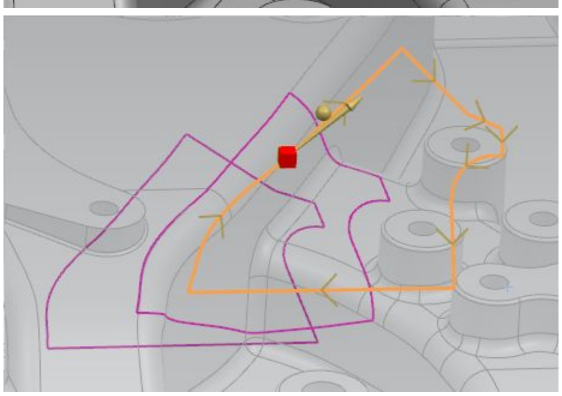
Boundary projection for Area Milling (10.0.1)

As you define or edit a boundary with the **Tool Position** option set to **Contact**, the outer boundary of the contact position is dynamically calculated and shown on all surfaces through the part.

The following images show arbitrary curves used to define a boundary that contains the cutter within a small area of the part.



The boundary curves contain the toolpath in the area shown.



When the **Trim Boundaries** dialog box is open, and the **Tool Position** option is set to contact, NX displays the tool contact boundary on all part surfaces. The boundary that is displayed on the surface that is being cut shows you the actual cut area.

Where do I find it?

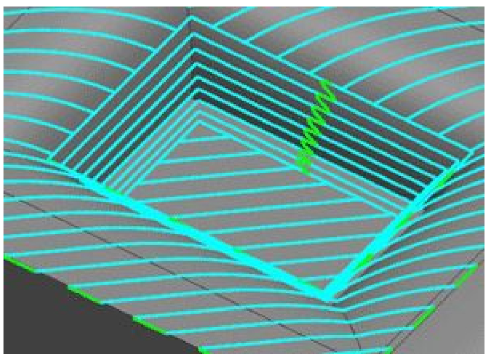
Initial release	NX 9.0.3 MP1
Application	Manufacturing

Prerequisites	Operations must use the **Area Milling** drive method. In geometry groups, the **Geometry Subtype** option must be set to **MILL_AREA**. In the **Trim Boundaries** dialog box, in the **Boundaries** group, the **Tool Position** option must be set to **Contact**.

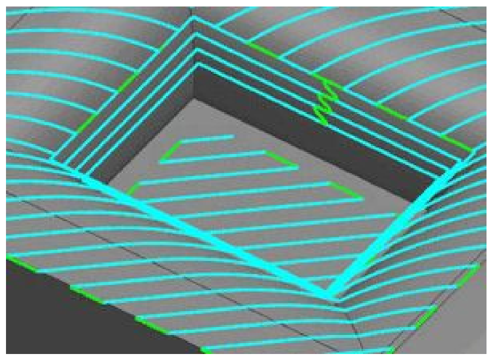
Minimum cut length for Area Milling (10.0.1)

What is it?

In Area Milling operations, you can now specify a minimum cut length for the steep Zlevel pattern.



Minimum cut length: 0.000”



Minimum cut length: 2.300”

Use the **Minimum Cut Length** option to determine which toolpaths to keep in the current operation. NX removes tool path cutting motions that are smaller than your specified minimum cut length, and cuts the Zlevel pattern with no holes or gaps.

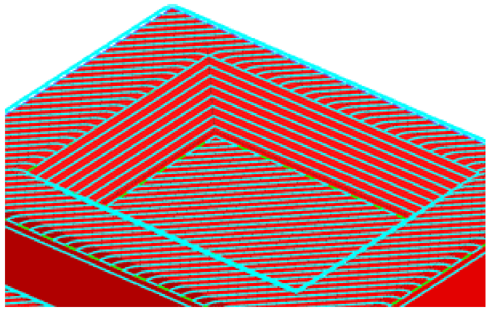
Where do I find it?

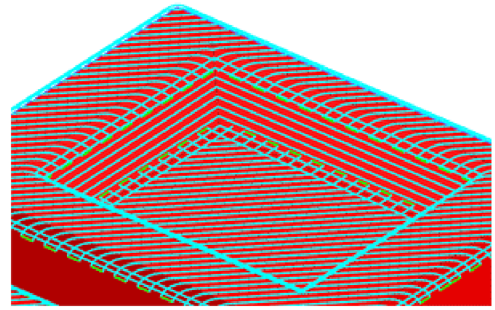
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Area Milling operation
Location in dialog box	**Area Milling Drive Method** dialog box→**Drive Settings** group→**Steep Cutting**

Cut region overlap in Area Milling (10.0.1)

What is it?

You can overlap the cutting motions between steep and non-steep regions in Area Milling operations. Use the **Overlap Steep and Non-Steep Regions** option to specify the distance by which the regions overlap.


Zero overlap applied


A .050” overlap applied

In the example shown, a .050” overlap has been applied. The .050” distance is divided by 2 and the regions are extended towards each other by .025” while following the part geometry.

Why should I use it?

When you overlap the cut regions, you can sometimes achieve a smoother finish.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	**Area Milling** operation
Location in dialog box	**Area Milling Drive Method** dialog box →**Steep Containment** group →**Overlap Steep and Non-steep Regions**

Divide a cut region using a line (10.0.1)

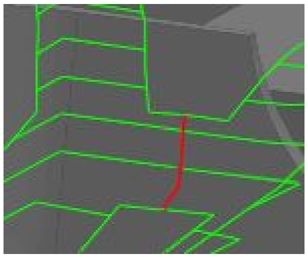
What is it?

In Area Milling operations, you can divide a cut region using a line with additional options. Use the Divide by Line option to create and control the placement of the line. You can:

- Create points that NX uses to create the line.
- Project the line on to the part.
- Preview the division. If the regions are not what you expect, you can change the line.

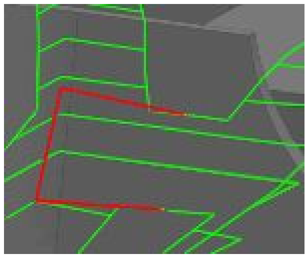
After you create the line, NX can create a plane that slices through the line and divides the cut region.

To specify how you want the plane to be oriented, use one of the following **Divide Plane** options:



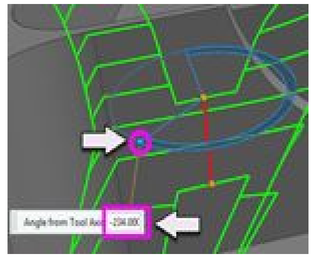
Tool Axis

Use this option to project the plane is projected through the current tool axis.



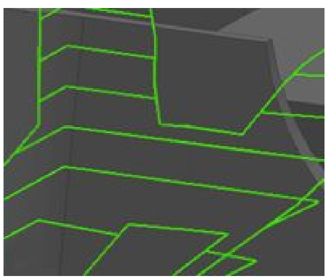
Normal to Tool Axis

Use this option to project the plane perpendicular to the tool axis.

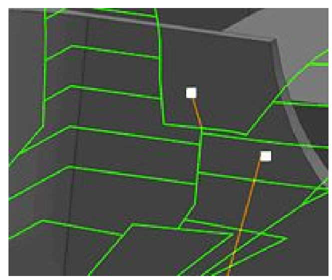


Dynamic

Use this option if you want to project the plane using an angle that you specify. You can either enter a value or drag the angel handle. The angle is referenced from the tool axis.



The cut region before any divisions.



The cut region after using the **Tool Axis Divide Plane** option.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Area Milling operation
Location in dialog box	[Area Milling operation] **Cut Regions** dialog box

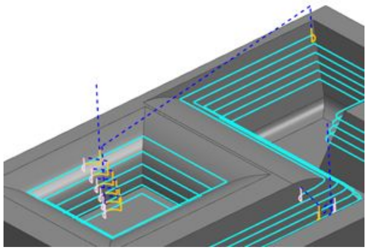
Creating gouge free traverse moves in Zlevel operations (10.0.1)

What is it?

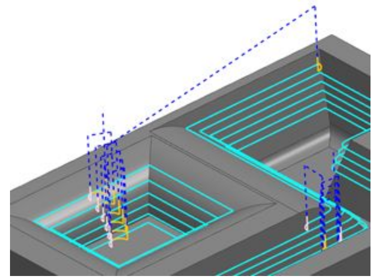
In Zlevel operations, for traverse moves, you can now avoid potential collisions between the cutter and the part or IPW. To do this, you set a safe clearance value and then apply it.

- To set the value, use the **Safe Clearance Distance** option.
- To apply the value, use the **Apply Safety Clearance to Direct** option.

If you do not set a **Safe Clearance Distance** value, NX notifies you that a potential collision could occur.



☐ Apply Safety Clearance to Direct



☑ Apply Safety Clearance to Direct

A 0.300” distance is used.

Option name change

In the **Non Cutting Moves** dialog box, on the **Transfer/Rapid** tab, on the **Transfer Type** list, the **Lowest Safe Z** option is now named **Direct / Previous Plane Backup**.

Where do I find it?

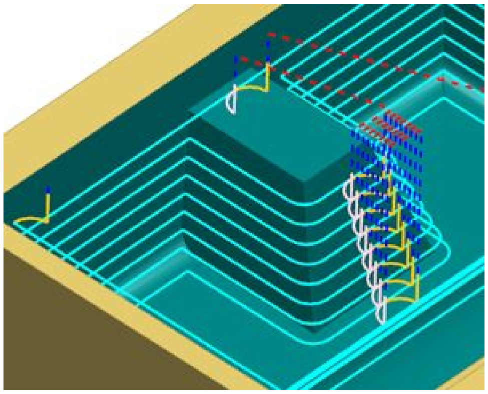
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Zlevel operation
Location in dialog box	[Zlevel operation]→**Path Settings**→**Non Cutting Moves**→**Transfer/Rapid** tab→**Between Regions** and **Within Regions** groups

Create smooth noncutting motions in Zlevel operations (10.0.1)

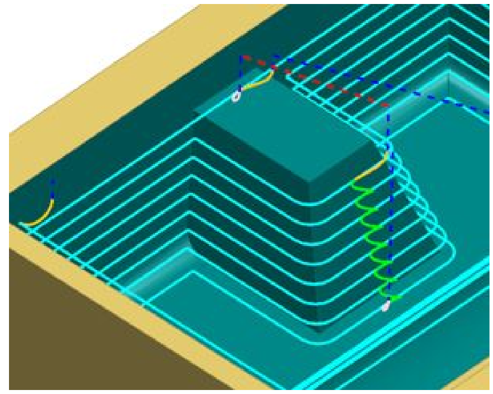
What is it?

You can override the engage, retract, and stepover motions within a region for Zlevel operations. To do this, use the **Override with Smooth Connections** option on the **Smoothing** tab. NX outputs a smooth continuous motion along a spline that is generated between cutting motions. You can specify the following parameters on the **Smoothing** tab:

- The length and height of the smooth engage and retract motions
- Maximum stepover and part safe clearance
- The distance of the smoothing region
- The tolerance



☐ Override with Smooth Connections



☑ Override with Smooth Connections

When machining electrodes for example, NX will apply safe clearances to Negative Part Stock values for traverse moves. NX also creates consistent, smooth stepover lifts off the part for each pass. If a smooth engage is not possible, NX notifies you and uses a plunge motion instead.

If you are not satisfied with the automatic start point of the tool path, you can specify the start point for the region on the **Start/Drill Points** tab.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Zlevel operation
Location in dialog box	**Non Cutting Moves** dialog box→**Smoothing** and **Start/Drill Points** tabs

Tilt Tool Axis enhancements (10.0.1)

What is it?

There are new usability enhancements that keep the latest tool axis settings stored in the following cases:

- If you generate a tool path using the **OK** command, and then stop it.
- If you generate a tool path and then use the **UNDO** command.

The **Tilt Tool Axis** dialog box now groups the options logically on the **Tilting**, **Clearance**, and **Machine Tool Characteristic** tabs.

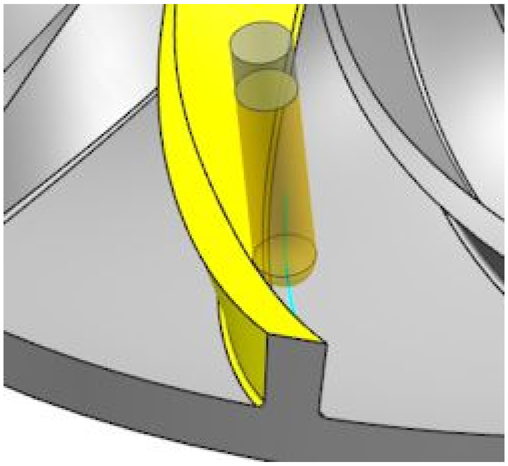
Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Command Finder	**Tilt Tool Axis**

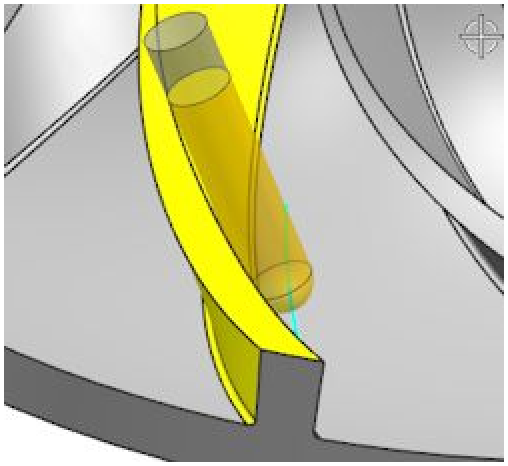
Swarf Blade tool axis enhancement

What is it?

For blade finish operations, the **Swarf Blade** tool axis option better aligns with blades that are appropriate for swarfing.



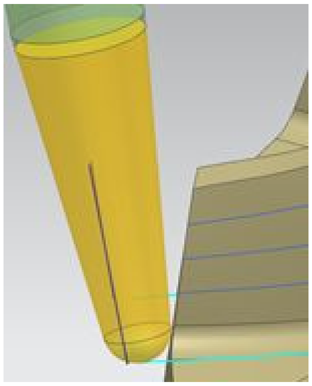
Previous versions



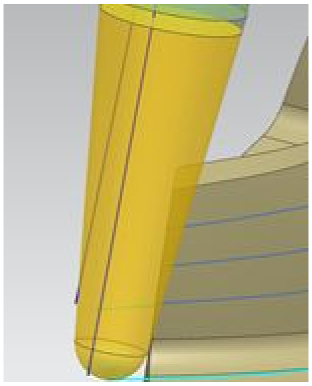
Current version

If the flute of your tool is not long enough to swarf cut the entire blade in a single pass, NX uses the same orientation for all of the cut levels.

Use the **Tilt Clearance Angle** option to specify a lead angle as the tool begins and ends the cutting pass. Only positive values are allowed.



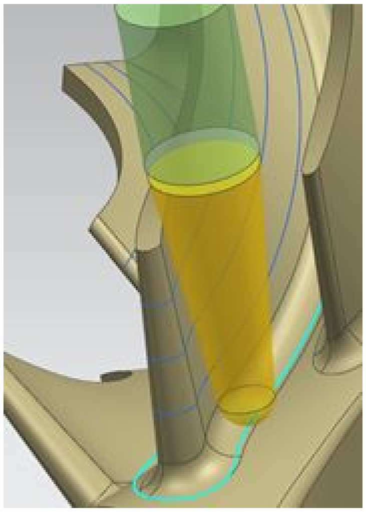
Tilt Clearance Angle = 20



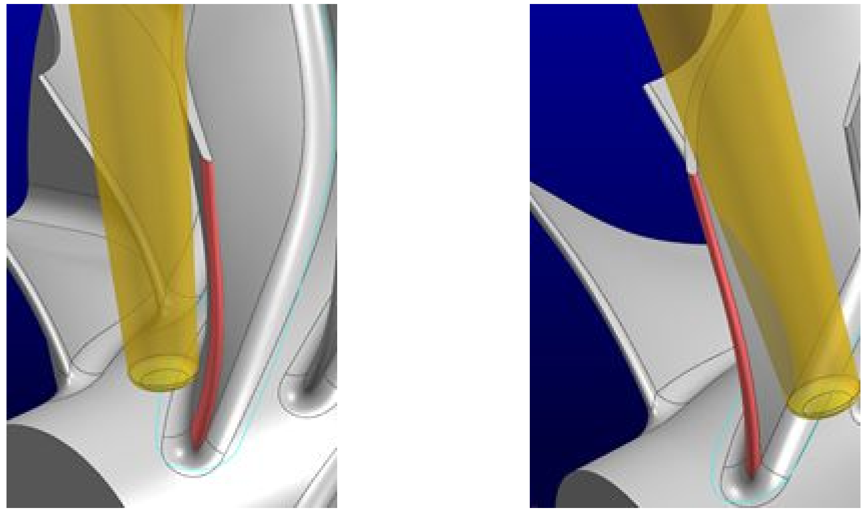
Tilt Clearance Angle = 0

Note

We recommend the **Swarf Blade** tool axis option for blades with ruled surfaces. It is not appropriate for blades with complex curvatures.



Ruled surface with perfect match



Complex curvature risks overcuts or gouges

Where do I find it?

Initial release	NX 9.0.1
Application	Manufacturing
Command Finder	**Create Operation**
Location in dialog box	**Create Operation** dialog box→**Type** list→**mill_multi_blade**→**Operation Subtype** group→**Blade Finish**

Completely machine both sides of a blade without edge rolling (10.0.1)

In multi blade operations, the **Blade Edge** option controls where the path stops following the blade geometry, and begins extensions that are not driven by the blade shape. You can now use the new **No Curling** edge trimming option to prevent the cutter from curling over the blade edges. This is helpful in the following cases:

- For blades with sharp corners at the trailing edge, the path follows the pressure and compression sides, and extends tangentially without any curling around the sharp corners.

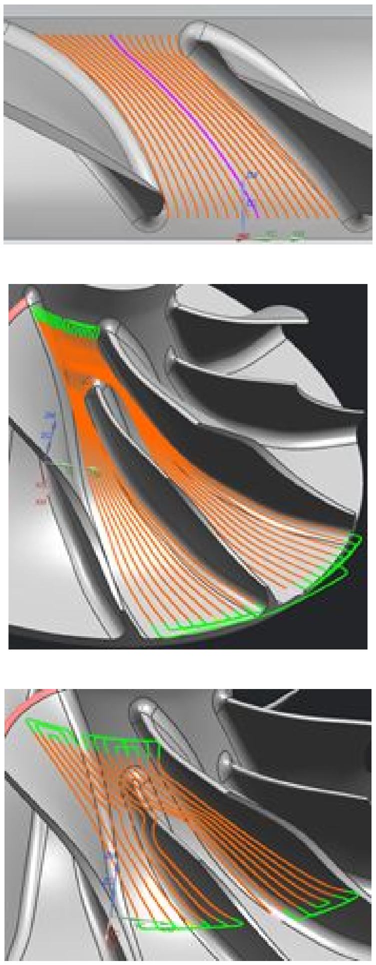
- For blades that are highly curved around the part axis, NX prevents fast changes in the path direction as the path moves from the pressure side to the leading edge.

- For operations with the cut levels depth mode set to **Offsets from Hub**, when a cut level reaches the shroud height, NX extends the path beyond the top edge of the blade without curling around the blade.

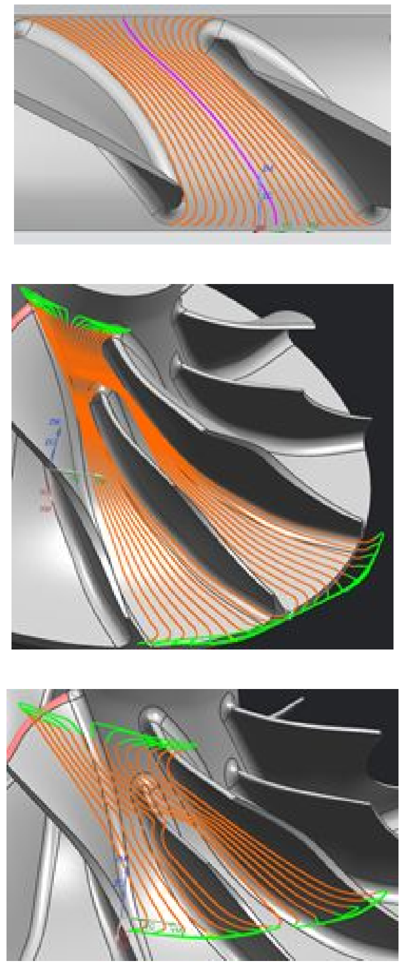
For roughing and hub finish operations, NX automatically adds a small tangential extension so that all passes of a cut level extend the same distance from the rotary axis.

For all operations, you can also enter an **Extension** value to extend the path tangentially. NX adds any user specified extension after the automatic extension.

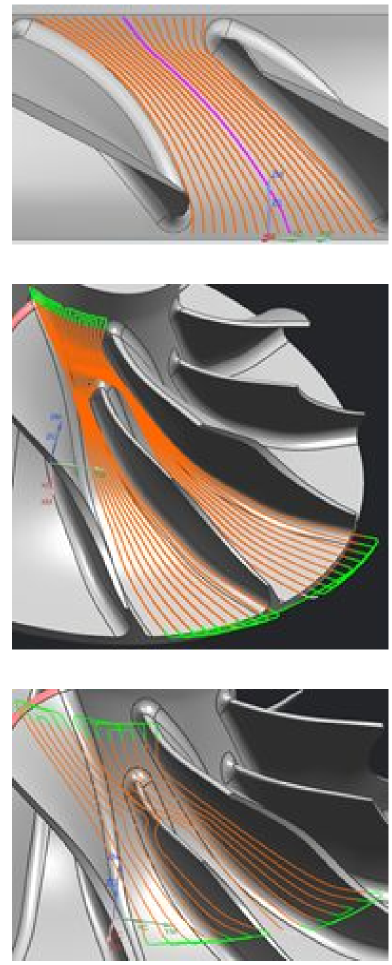
The following examples show how the **No Curling** option automatically controls the tool path at the leading and trailing blade edges. The stepovers are exaggerated to show detail.



Blade Edge = Along Blade Direction



Blade Edge = Along Part Axis



Blade Edge = No Curling

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Multi Blade operation
Location in dialog box	[Multi blade operation] dialog box→**Drive Method** group→edit drive method  Blade rough, hub finish operations: **Multi Blade Rough Drive Method** or **Hub Finish Drive Method** dialog box→**Leading Edge** group Blade finish operations: **Blade Finish Drive Method** dialog box→**Cut Periphery** group→**Leading Edge** subgroup Blend finish operations: **Blend Finish Drive Method** dialog box→**Cut Periphery** group→**Trailing Edge** subgroup

Allowed violations of part geometry (10.0.1)

In the path settings for an operation, when you check for gouges, NX normally reports a gouge and does not generate a gouging tool path.

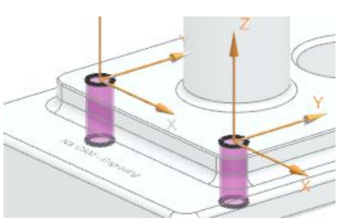
For some situations, you implicitly authorize a gouge to the modeled body by defining a machining feature that is not explicitly modeled. In these situations, NX generates a tool path and does not report gouges within a defined feature volume. For authorized gouges, NX gouge checking reports `No gouged motions found`.

The following are examples of implicitly authorized gouges.

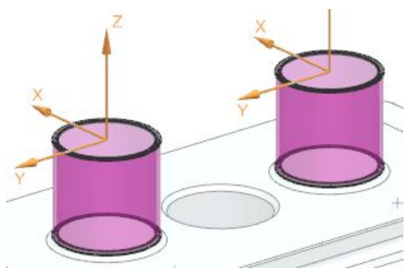
Tapping and thread milling

Typically, you model only the tap diameter of a hole or the maximum thread diameter of a boss. NX creates the feature volume during the machining feature recognition process.

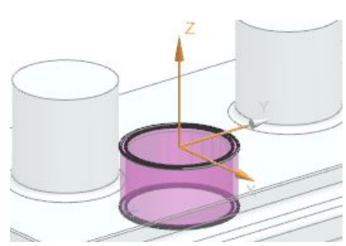
Tapped hole volumes



Thread mill boss volume



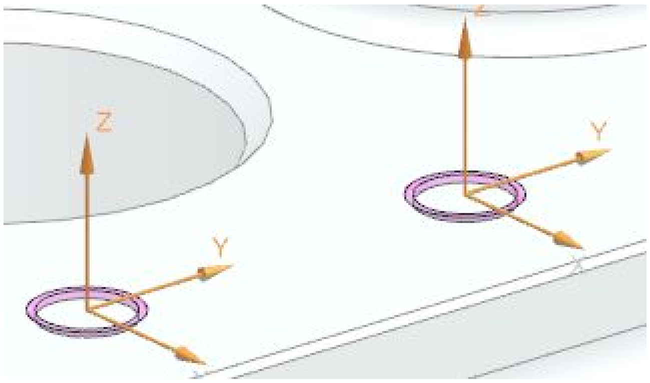
Thread mill hole volume



Chamfering holes

You need not model the chamfer.

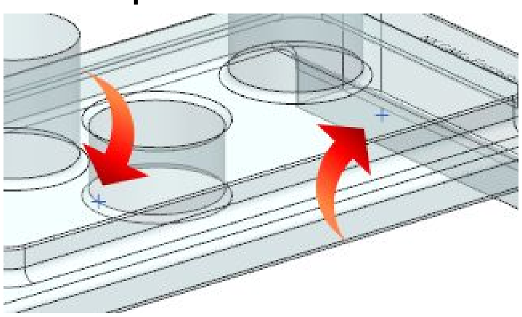
Chamfer volumes



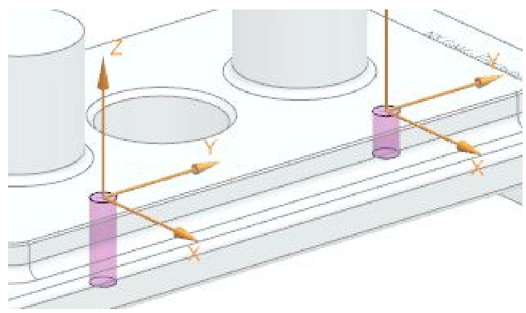
Drilling holes at points

You can specify a hole feature at a point, without modeling the hole.

Modeled points



Hole feature volumes

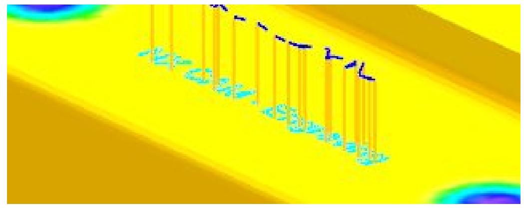


Engraving

When you engrave text using **Planar Text** and **Contour Text** operations, the text to be cut exists only as a 2D note. There is no corresponding feature volume, and the operation dialog box does not contain a gouge checking option. No gouge is reported when you run a gouge check.

2D drafting note

Tool path



Where do I find it?

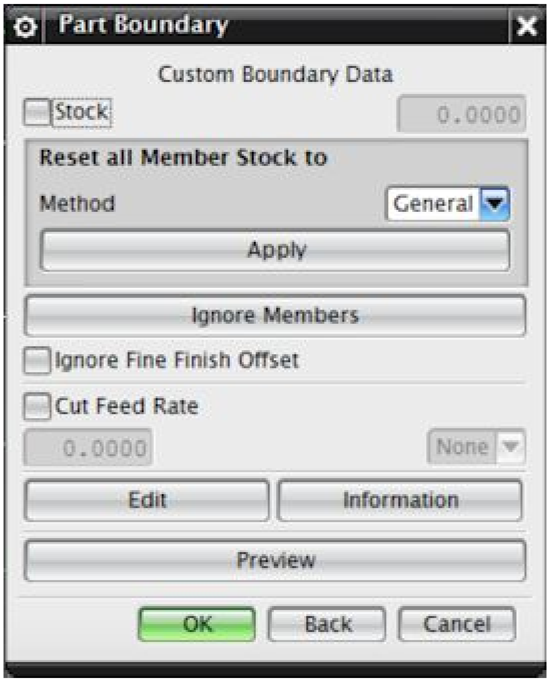
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Feature volumes must be defined for the operation.
Operation Navigator	Right-click operation→**Gouge Check** Right-click operation→**Tool Path**→**Verify**→**Replay**→**Gouge and Collision Settings**
Operation dialog box	**Generate** **Verify**→**Replay**→**Gouge and Collision Settings**
Tool Path Editor dialog box	**Analyze Tool Path** group→**Gouge and Collision Check**
Simulation Control Panel dialog box	For tapping and thread milling only: **Simulation Settings** group→**Simulation Settings** →**Simulation Settings** dialog box→**Collision Detection** group→**Specify Collision Pairs** →**Specify Collision Pairs** dialog box→specify the part and the tool

Turning

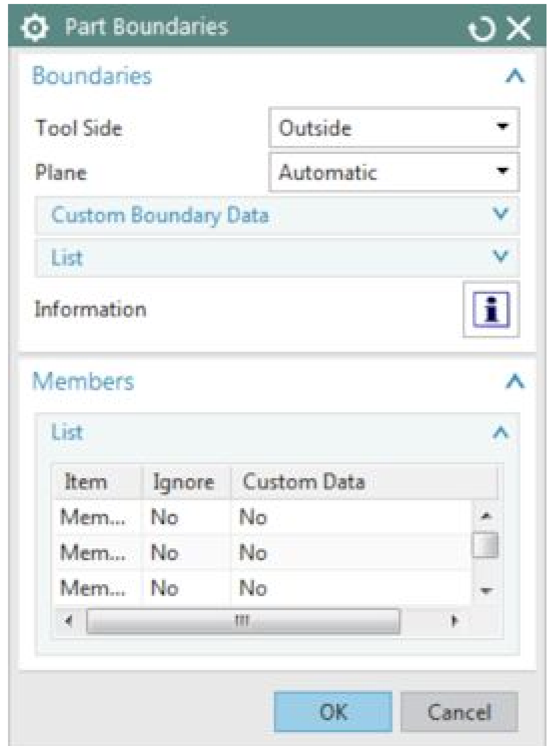
Part Boundary dialog box enhancements

The look of the dialog boxes where you select boundary options in NX turning has changed.

Pre-NX 9.0.2



NX 9.0.2



Turning workpiece

The names of boundary selection dialog boxes that you access from the **Turn Bnd** dialog box have changed. To display the **Turn Bnd** dialog box:

- In the Geometry view of the **Operation Navigator**, right-click **Turning Workpiece** and choose **Edit**.

Path to boundary selection dialog box	Pre-NX 9.0.2 name	NX 9.0.2 name
Click **Specify Part Boundaries** 	**Part Boundary**	**Part Boundaries**
Click **Specify Blank Boundaries** 	**Select Blank**	**Blank Boundaries**

Turning operation

The name of boundary selection dialog boxes that you access from a turning operation have changed.

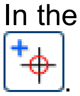
- Right-click any turning operation and choose **Edit**.

Path to boundary selection dialog box	Pre-NX 9.0.2 name	NX 9.0.2 name
Geometry group→**Custom Part Boundary Data**	**Part Boundary**	**Part Boundaries**

Teach Mode operation

The names of boundary selection dialog boxes that you access from a **Profile Move** type of **Teach Mode** operation have changed.

1. Create a **Teach Mode** operation.

2. In the **Teach Mode** dialog box, in the **Sub-Operations** group, click **Add New Sub-Operation** .

3. In the **Create Teachmode Subop** dialog box, from the **Move Type** list, select **Profile Move**.

Path to boundary selection dialog box	Pre-NX 9.0.2 name	NX 9.0.2 name
Path Settings group→**Drive Geometry** list→**New Drive Curve**	**Select Drive Geometry**	**Part Boundaries**
Start and Stop group→**Start Position** list→**Check Curve**→**Start Check Geometry**	**Select Start Check Geometry**	**Start Check Boundary**
Start and Stop group→**Start Position** list→**Check Curve**→**Stop Check Geometry**	**Select Stop Check Geometry**	**End Check Boundary**

Where do I find it?

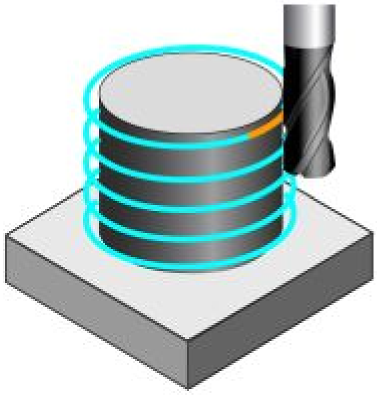
Initial release	NX 9.0.2
Application	Manufacturing

Hole machining

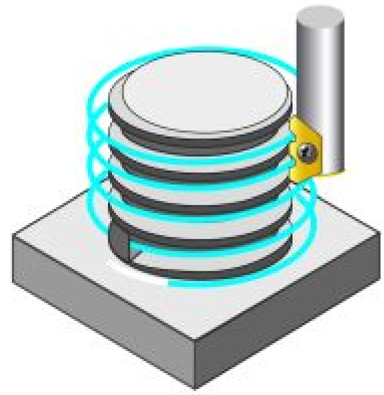
Boss milling and threading operations

What is it?

There are new operations to mill and thread boss geometry. You can select only boss geometry from within these new operations.



Boss Milling



Boss Thread Milling

The following operations now machine only hole geometry.

- **Hole Milling**
- **Thread Milling**

The **Specify Hole or Boss** button is now called **Specify Feature Geometry**. Within each operation you can select or edit only the appropriate feature geometry.

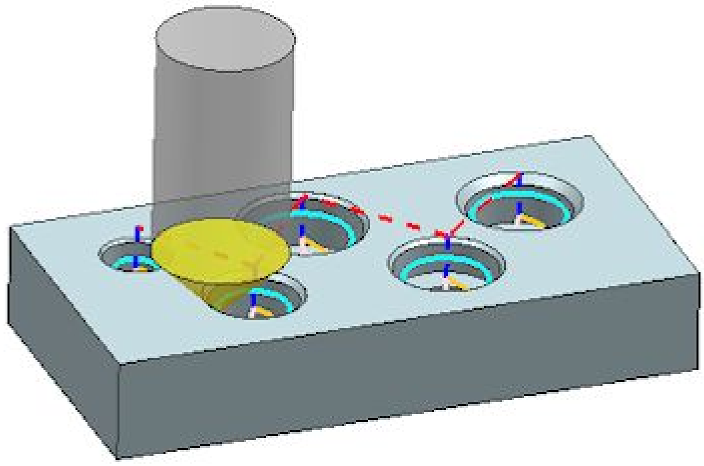
Why should I use it?

The new operation types make it easier to identify which operations mill holes and which operations mill bosses.

Where do I find it?

Initial release	NX 9.0.1
Application	Manufacturing
Command Finder	**Create Operation**
Location in dialog box	**Create Operation** dialog box→**Type** list→**hole_making**→**Operation Subtype** group→**Boss Milling** or **Boss Thread Milling**

Milling chamfers on holes



Mill hole chamfers using the **Hole Chamfer Milling** operation to cut a single chamfer on multiple features.

- Select hole features to chamfer, or select geometry from the graphics window. You do not need a modeled chamfer.

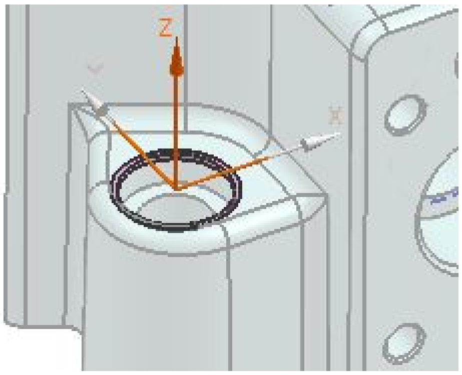
- For modeled chamfers, select chamfer machining areas to chamfer, or select geometry from the graphics window. For unmodeled chamfers, select the cylindrical face to chamfer and specify the counter sink diameter.

- When the angle of the cutter tip does not match the angle of the modeled chamfer:

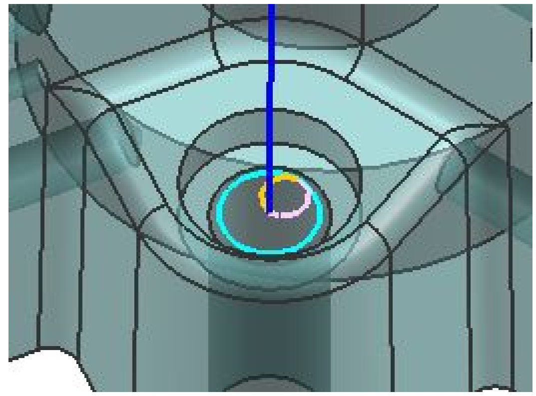
  o The operation leaves material if the cutter angle is less than the chamfer angle.

  o The tool gouges if the cutter angle is greater than the chamfer angle. NX reports the gouges.

The operation determines the in-process feature volume for the chamfer and creates a single circular cut around the hole diameter.



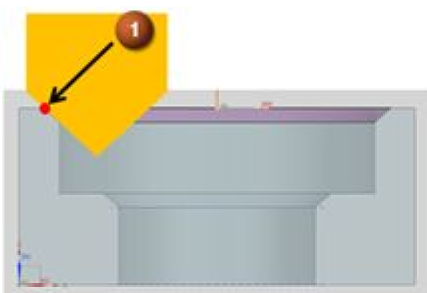
Chamfer in-process feature volume



Chamfer tool path

Tracking points

NX drives the tracking point you select as the drive point along the top edge of the chamfer. Usually the tracking point is located on the cutting edge, but NX can use any tracking point that you select.



1 Tracking point on spot drilling tool used as the drive point in the operation

Note

Because the operation cuts chamfers on holes with or without a modeled chamfer, NX uses the selected tracking point even if the tool gouges the part or cuts air.

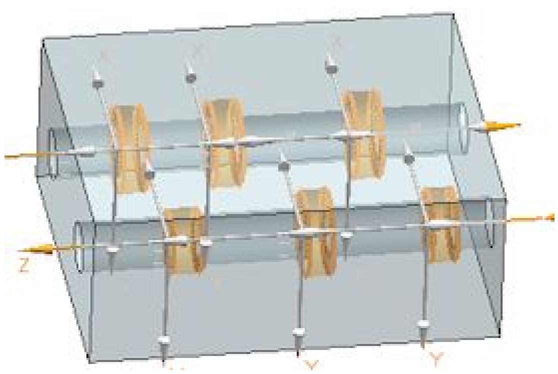
Contact and tracking data output

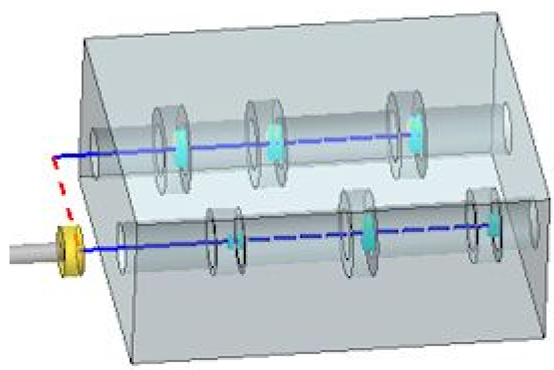
NX optionally outputs contact/tracking data at the tracking point defined on the tool diameter. Otherwise, NX outputs the tool path at the tool tip centerline.

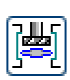
Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	A supported chamfering tool such as a chamfer mill, spot drill, or countersink tool. The tool must have the following attribute: • Defined tracking point.
Command Finder	**Create Operation**
Location in dialog box	Hole Chamfer milling operation **Create Operation** dialog box→**Type** list→**hole_making**→**Hole Chamfer Milling**  Selecting the tracking point **Hole Chamfer Milling** dialog box→**Path Settings** group→**Drive Point** list

Radial Groove Milling (10.0.1)





The **Radial Groove Milling**  operation combines groove milling and hole milling to machine a sequence of circular grooves with a T-cutter. The operation cuts in a circular pattern with multiple radial and axial passes.

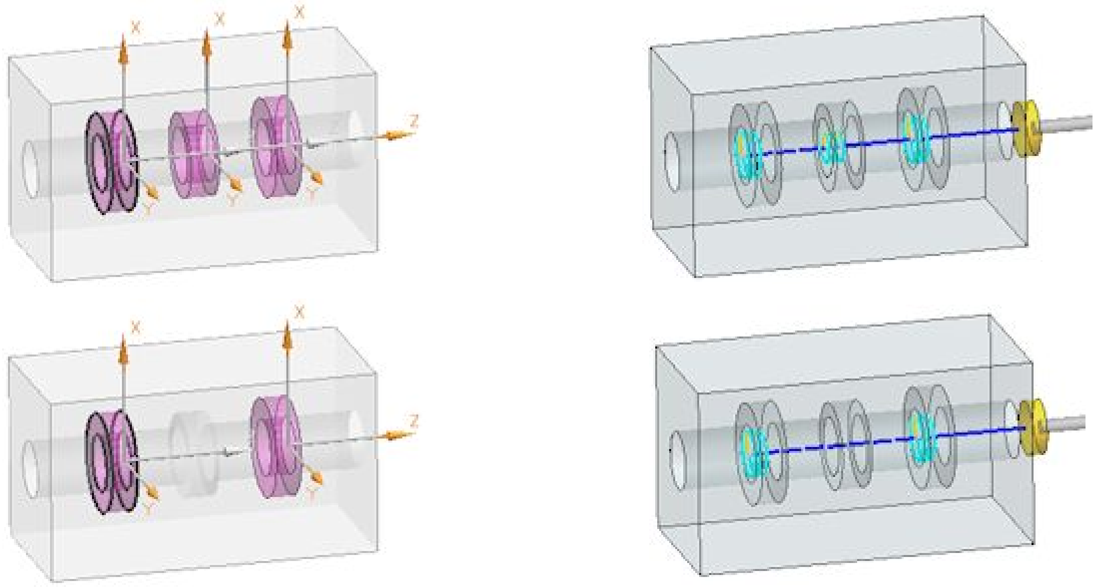
Your workflow can include multiple operations. Create a roughing operation that leaves stock on the floor, ceiling, and side, and then remove the remaining material in a subsequent finishing operation.

You can manually select the groove geometry from within the operation, or use previously recognized groove features within a feature group. NX updates the tool path when you regenerate if there are:

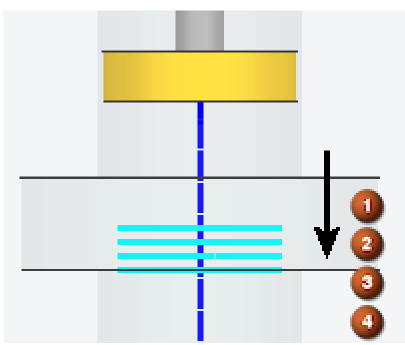
• Dimensional changes

- Sequence changes

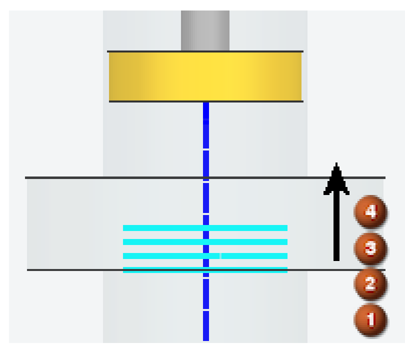
  You can add grooves to the sequence or remove grooves from the sequence.



The **Radial Groove Milling** operation uses the same level sequencing options that are available in **Groove Milling**. For **Radial Groove Milling** operations, the ceiling is the face closest to the direction that the tool approaches from.



Ceiling to Floor

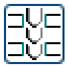


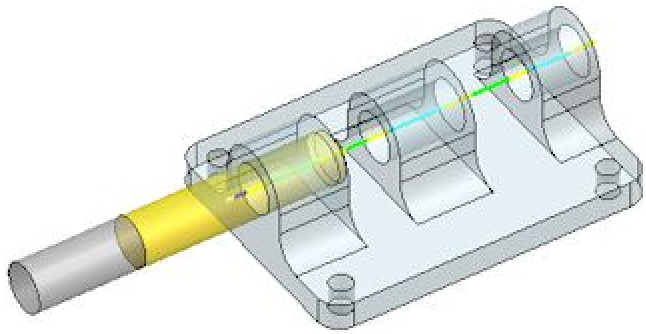
Floor to Ceiling

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	T-cutter tool, enclosed groove
Command Finder	**Create Operation**
Location in dialog box	**Create Operation** dialog box→**Type** list→**hole_making**→**Operation Subtype** group→**Radial Groove Milling**

Drilling interrupted holes (10.0.1)

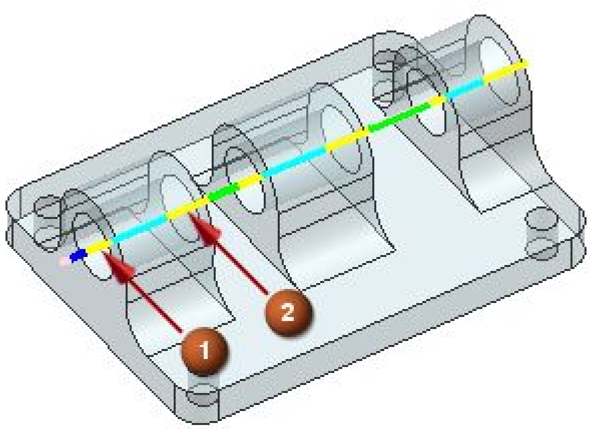
Use the **Sequential Drilling**  operation to drill a sequence of coaxial holes.



You can:

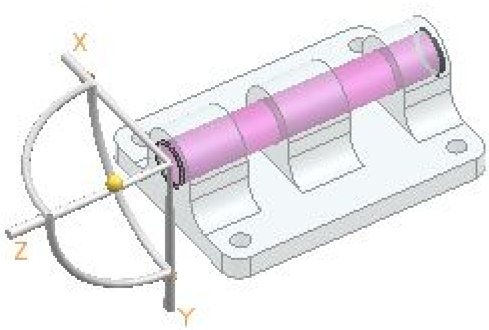
- Create a feature group geometry parent first, or specify the geometry from within the operation.
- Set the cycle tracking point.
- Drill with or without chip break.

  For the chip break cycle, you can also specify a clearance and retract for each hole segment.
- Slow the feed rate when the tool enters material, and again when the tool breaks through material. Use the **First Cut** and **Last Cut** options to specify the distance and feed rates for entering (1) and leaving (2) material.



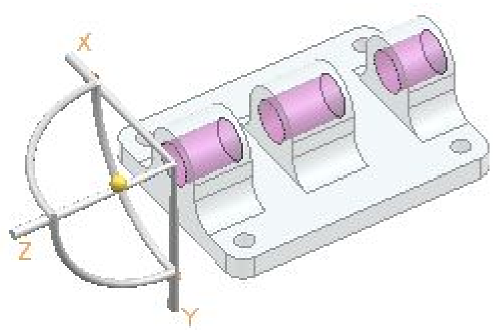
Interrupted hole features

For a **Sequential Drilling** operation, the **Motion Output** option is set to **Interrupted Moves**. This tells NX to ignore empty spaces when it creates the feature geometry.



Hole feature

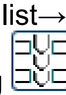
Motion Output = **Machine Cycle**



Hole feature

Motion Output = **Interrupted Moves**

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Command Finder	**Create Operation** 
Location in dialog box	**Create Operation** dialog box→**Type** list→**hole_making**→**Operation Subtype** group→**Sequential Drilling** 

Machining through holes and blind holes together

You can group through holes and blind holes to machine them together. You save on programming time because you do not have to program the through holes and blind holes separately. Use the **Predefined** option of the **Group Features** command to group features that have identical diameters, thread pitch, and number of steps. When it groups the features, NX ignores the values of all depth attributes, including tolerances, chamfers and thread lengths.

How it works

In NX, through holes are **STEPxHOLE** features and blind holes are **STEPxPOCKET** features, where x indicates the number of steps in the feature. When you group features using the **Predefined** option, NX:

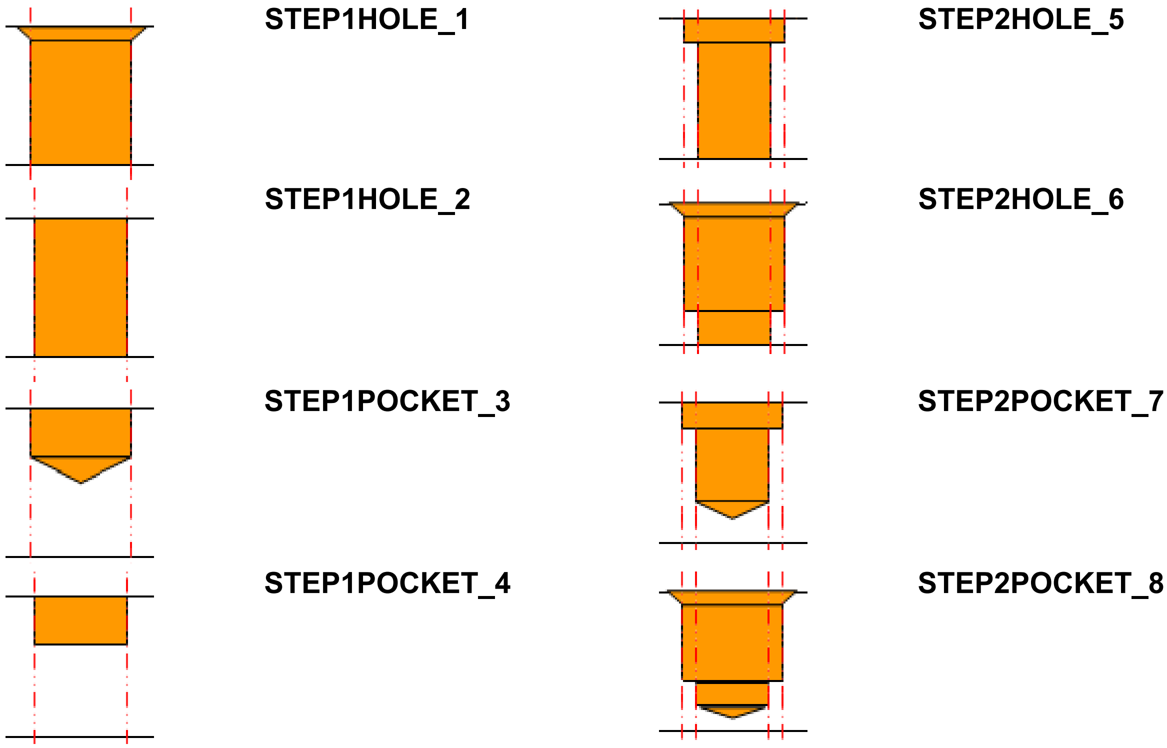
- Includes all **STEP1HOLE** and **STEP1POCKET** features with identical diameters in an **FG_STEP1HOLE_STEP1POCKET** feature group.
- Includes all **STEP1HOLE_THREAD** and **STEP1POCKET_THREAD** features with identical diameters and thread pitch in an **FG_STEP1HOLE_THREAD_STEP1POCKET_THREAD** feature group.

- Repeats the process for features with multiple steps.

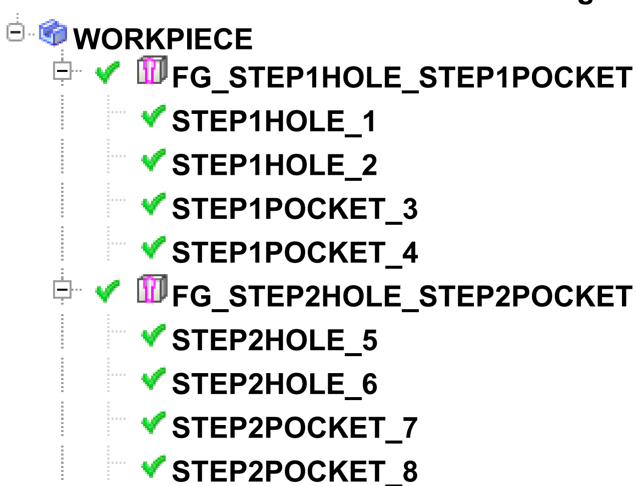
If additional feature groups are required for holes with different diameters, NX creates the feature groups and names them **FG_STEP1HOLE_STEP1POCKET_1**, **FG_STEP1HOLE_STEP1POCKET_2** and so on.

The following example shows different hole feature profiles, and how NX groups the hole features together.

Features to group

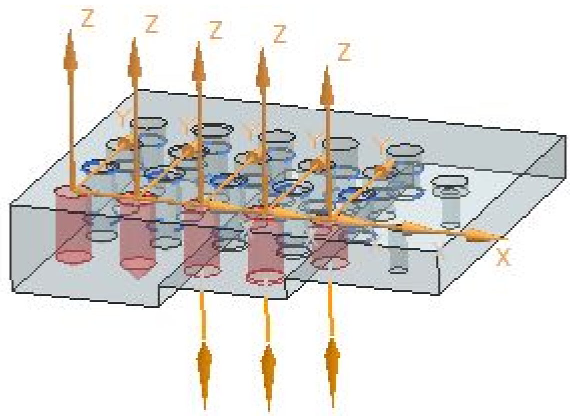


Features in the Machining Feature Navigator after grouping

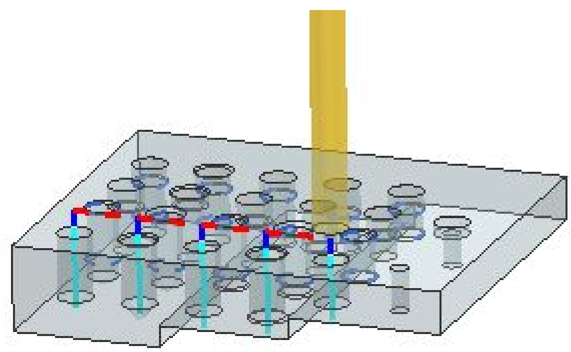


Sample part

Features selected for **FG_STEP1HOLE_STEP1POCKET**



Drilling operation for **FG_STEP1HOLE_STEP1POCKET**



The **Group Features** command also has the following options:

- **None** groups features whether or not the attributes are identical.
- **All** groups features only if all of the attributes are identical.

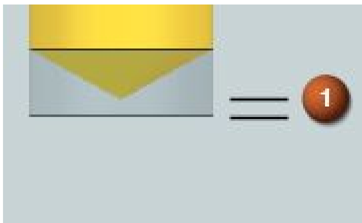
Where do I find it?

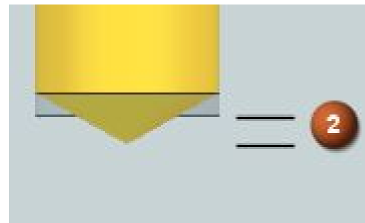
Initial release	NX 9.0.2
Application	Manufacturing
Machining Feature Navigator	Right-click in the background→**Group Features** Grouping selected features Right-click selected features→**Group Features**
Location in dialog box	**Group by** group→**Identical Attributes** list →**Predefined**

Enhancements to control drilling depth

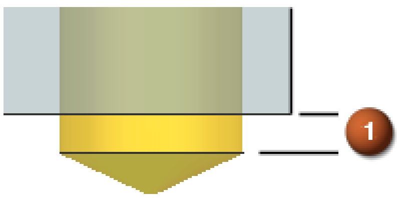
You can control drilling depth for drills and step drills in the following ways.

- Apply a positive (1) or negative (2) bottom stock to blind holes.





- Apply a bottom offset (1) to through holes.



- Use the **MODEL_DEPTH** option to drill to the depth where the tool contacts the part.

  Use this method to control cutting depth for hole features where depth attributes are not relevant or not desired.

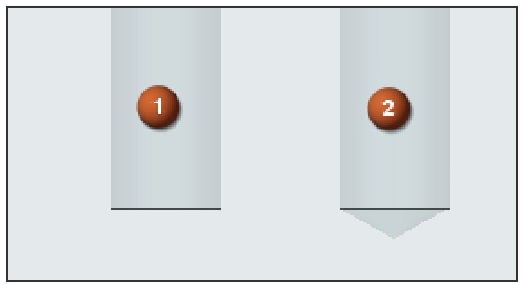
- Use the **Position Tool Shoulder** option to control whether the tool contact point is at the tool tip or the tool shoulder.

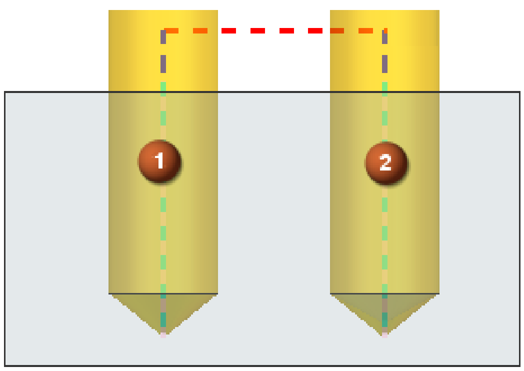
  Note

  The **Position Tool Shoulder** option is separate from the **MODEL_DEPTH** option.

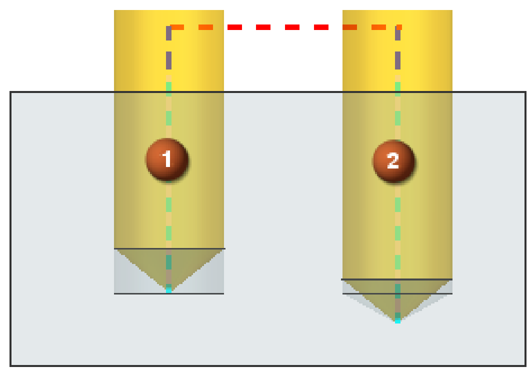
Drilling to tool contact depth

Use the **MODEL_DEPTH** option to drill to where the tool contacts the part. You control the drilling depth by choosing whether the tool contact point is at the tool tip or the tool shoulder. In the following example, (1) is a hole modeled with a flat bottom and (2) is a hole modeled with a tip angle.





Position Tool Shoulder = ☑



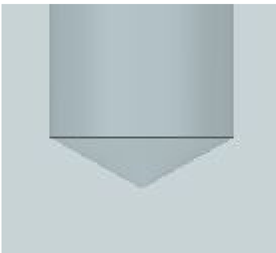
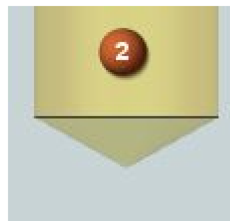
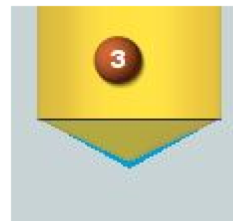
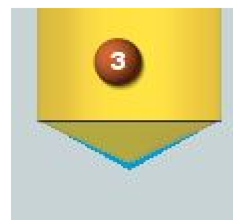
Position Tool Shoulder = ☐

If you drill the hole with a tool that does not match the tip angle of the modeled hole, the operation will either leave material or gouge the part.

- If you use the tool shoulder to position the tool, and the drill tip angle is less than the modeled tip angle, the operation gouges the part.

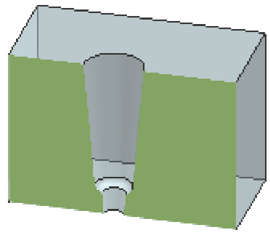
- If you use the tool tip to position the tool, and the drill tip angle is greater than the modeled tip angle, the operation drills as deeply as it can without gouging the part.

The following example shows a hole feature modeled with a 122° tip.

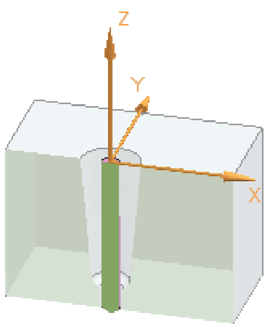
Hole	102° drill tip	122° drill tip	130° drill tip
			
Position to tool shoulder			
	Gouges	Exact match	Leaves material
Position to tool tip			
	Leaves material	Exact match	Leaves material

Conical holes

Use the **Model Depth** and **Bottom Stock** options to help you drill conical holes. The following example shows the in-process features for the operations used to rough a tapered hole.



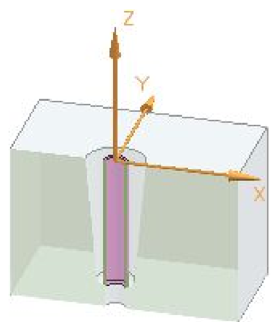
Volume to drill: 5 mm through hole with conical taper from 8 mm to 13 mm



4 mm drill

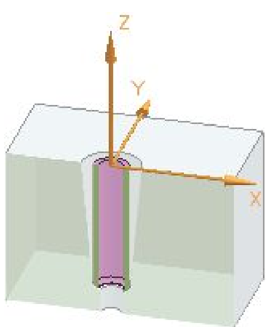
Bottom Stock = 0

Bottom Offset = 1.0



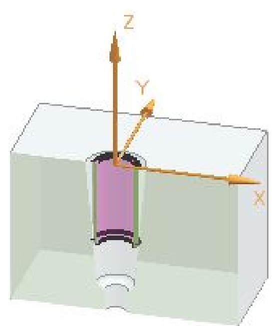
5.5 mm drill

Bottom Stock = 0.1



8 mm drill

Bottom Stock = 0.1



10 mm drill

Bottom Stock = 1.0

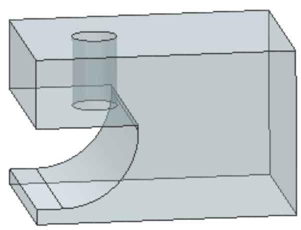
Protecting part geometry below a through hole

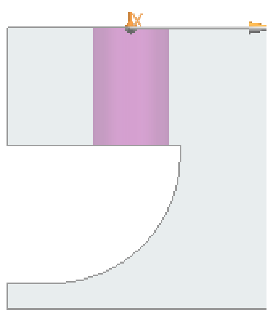
Use the **Model Depth** and **Bottom Stock** options also for through holes when you need to extend the depth of the in-process feature to the part geometry.

Note

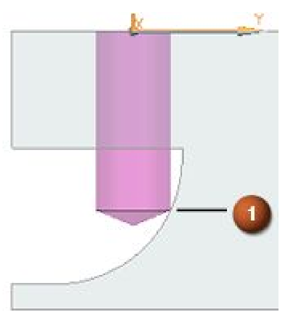
> When you select the **Model Depth** feature geometry setting, and the in-process feature of a through hole extends to the part geometry, NX ignores the **Bottom Offset** cutting parameter setting. NX uses the **Bottom Stock** setting instead.

The following example shows the in-process features and resulting tool path for different **Machining Area** and **Bottom Stock** option settings.





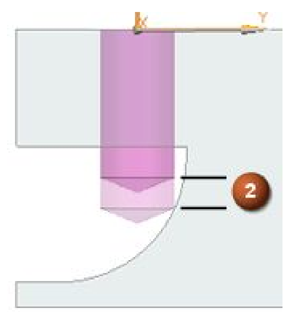
Machining Area = **FACES_CYLINDER_1**



Machining Area = **MODEL_DEPTH**

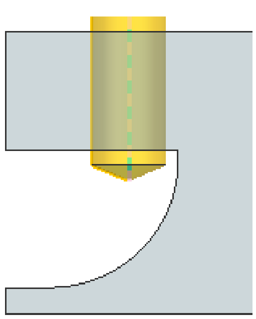
1 **Bottom Stock** = 0

(contact point)

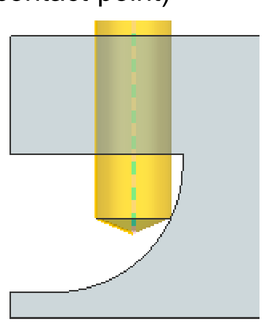


Machining Area = **MODEL_DEPTH**

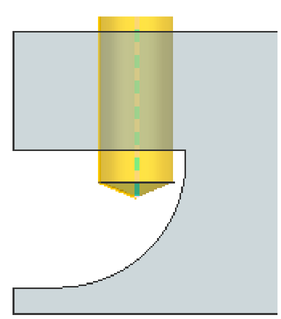
2 **Bottom Stock** = 5



Bottom Offset = 2



Bottom Offset = 2



Bottom Offset = 2

Where do I find it?

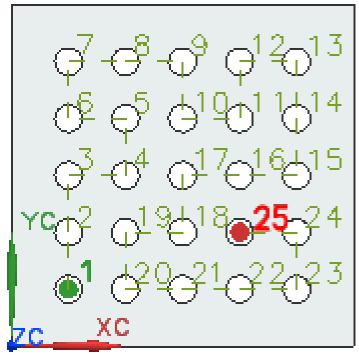
Initial release	NX 9.0.2
Application	Manufacturing
Location in dialog box	Drilling to model depth **Feature Geometry** dialog box→**Common Parameters** group→**Machining Area** list →**MODEL_DEPTH** Positioning to the tool shoulder or tool tip **Feature Geometry** dialog box→**Common Parameters** group→**Cutting Parameters** subgroup→**Position Tool Shoulder** Adding bottom stock to blind holes **Feature Geometry** dialog box→**Common Parameters** group→**Cutting Parameters** subgroup→**Bottom Stock** Adding bottom offset to through holes [Drilling operation] dialog box→**Cutting Parameters**→**Cutting Parameters** dialog box→**Strategy** tab→**Extend Path** group→**Bottom Offset**

Enhancements to optimize drill sequencing

You can optimize the drilling sequence in the following ways:

- Machine parts with holes arranged in a grid using a zig or zig-zag pattern to minimize tool travel. Whenever possible, NX keeps the transition motions parallel to the axis or vector that you specify as the primary pattern direction.
- Reverse the order of the listed hole or boss locations.
- Select a hole feature and make it the start or end location of the cutting sequence.

To see the machining order without generating the operation, preview the hole sequence.

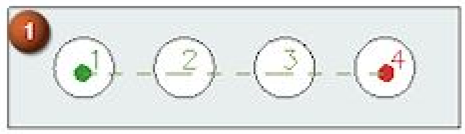

7
8
9
12
13
6
5
10
11
14
3
4
17
16
15
YC
2
19
18
25
24
1
20
21
22
23
ZC
XC

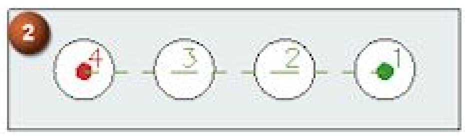
Defining the hole sequence

You can use the hole sequence defined in the **HOLE_BOSS_GEOM** geometry parent, or modify the sequence for an individual operation.

Operation Navigator — Geometry

- **WORKPIECE**
  - **HOLE_BOSS_GEOM** Defines the default hole sequence (1).
    - **SPOT_DRILLING** Uses the default sequence.
    - **DRILLING** Modified to use a reversed hole sequence (2).
    - **COUNTERSINKING** Uses the default sequence.





Use the **Reload List From Parent** sequencing option in the operation after changes in the parent sequence.

Note

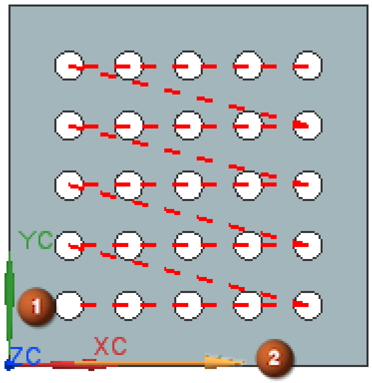
The options in the **Sequence** group in the **Hole or Boss Geometry** and **Feature Geometry** dialog boxes always display the default settings. The sequencing options are *actions* that NX execute when you click **Reorder List**. NX saves the resulting sequence without saving the settings.

Machining holes using a zig or zig-zag pattern

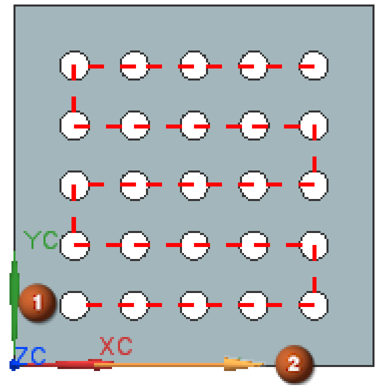
To minimize tool travel, machine parts with holes arranged in a grid using a zig or zig-zag pattern. Whenever possible, NX keeps the transition motions parallel to the axis or vector that you specify as the primary pattern direction.

The following example compares a zig pattern and a zig-zag pattern with the same start (1) point. The primary pattern direction (2) is along the X-axis.

Zig pattern

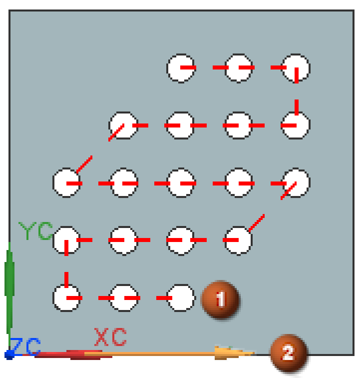


Zig-zag pattern

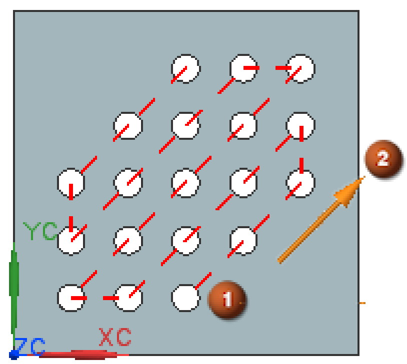


The following example compares two zig-zag patterns with the same start (1) point, but different primary pattern directions (2).

Primary direction along X-axis



Primary direction along specified vector



Changing the start or end points

To make a feature the start or end location, select it and click **Move to top** or **Move to Bottom**.

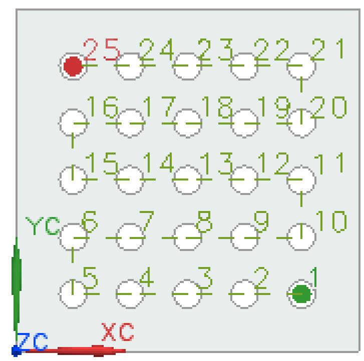
Caution

Specifying an end location when using a **Zig** or **Zig-zag** pattern is not recommended. NX determines the appropriate end location for you.

The following example compares the **Hole or Boss Geometry** dialog box list and preview for a hole sequence before and after changing the start location.

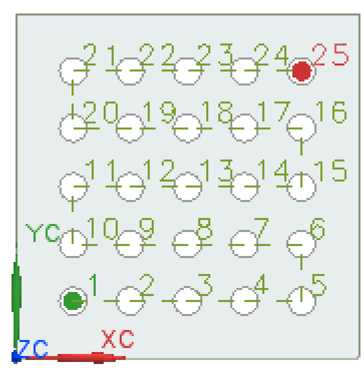
Hole sequence with **STEP1HOLE_1** as the start location.

Item	Name
1	STEP1HOLE_1
2	STEP1HOLE_2
3	STEP1HOLE_3
4	STEP1HOLE_4
5	STEP1HOLE_5
.	.
.	.
.	.
25	STEP1HOLE_25



Hole sequence with the start location changed to **STEP1HOLE_5**. Note that NX retains the original names of the holes.

Item	Name
1	STEP1HOLE_5
2	STEP1HOLE_4
3	STEP1HOLE_3
4	STEP1HOLE_2
5	STEP1HOLE_1
.	.
.	.
.	.
25	STEP1HOLE_21

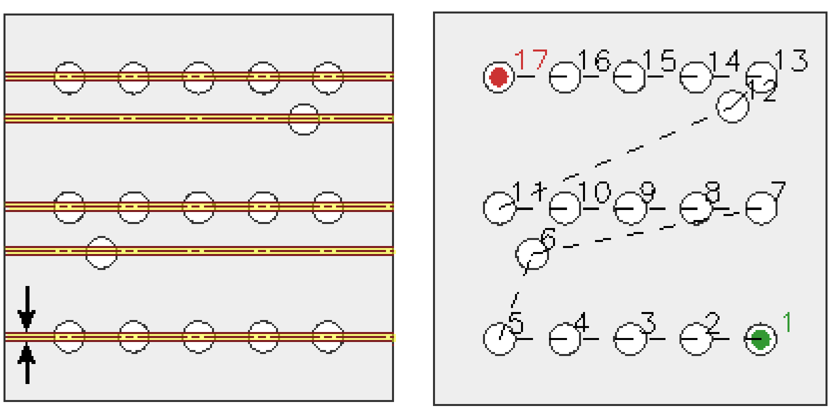


Where do I find it?

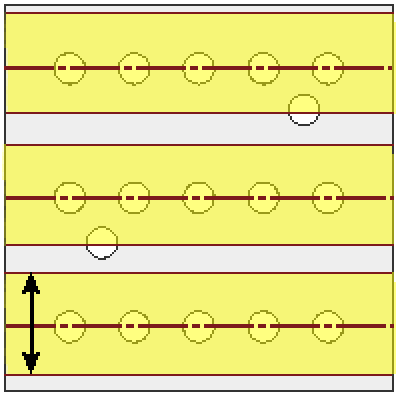
Initial release	NX 9.0.2
Application	Manufacturing
Location in dialog box	Defining the sequence for multiple operations **Hole or Boss Geometry** dialog box Defining the sequence for a single operation [Drilling operation] dialog box→**Geometry** group→**Specify Feature Geometry**→**Feature Geometry** dialog box Changing the sequence start or end location **Hole or Boss Geometry** or **Feature Geometry** dialog box→**Feature** group→**List** subgroup→**Move to top** or **Move to Bottom** Zig or zig-zag pattern sequence **Hole or Boss Geometry** or **Feature Geometry** dialog box→**Sequence** group→**Optimization**→**Primary Direction**

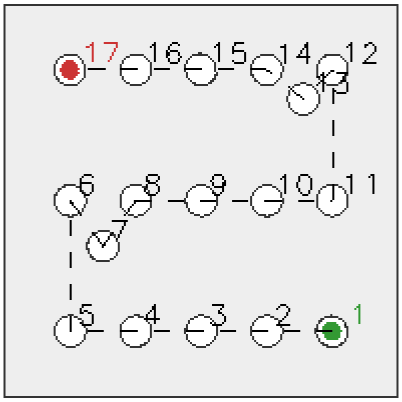
Controlling the width for zig and zig zag hole patterns (10.0.1)

When you optimize a machining sequence, the default tolerance that NX uses for zig and zig zag hole patterns can create additional passes when the holes are not aligned. Use the **Band Width** option to specify a width that is large enough to include nearby holes in each pass. NX applies half of the specified value to both sides of the pass center line.



Band Width = 1 mm (default value)

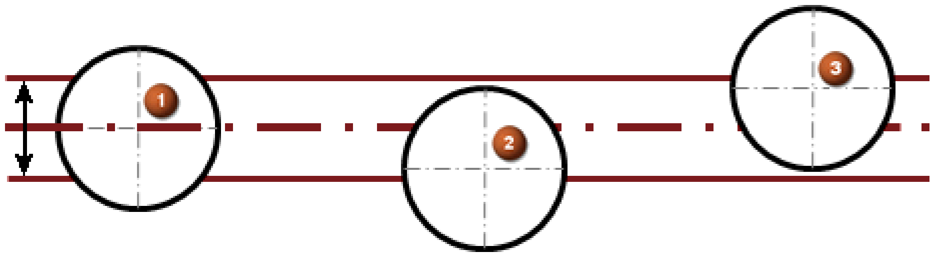




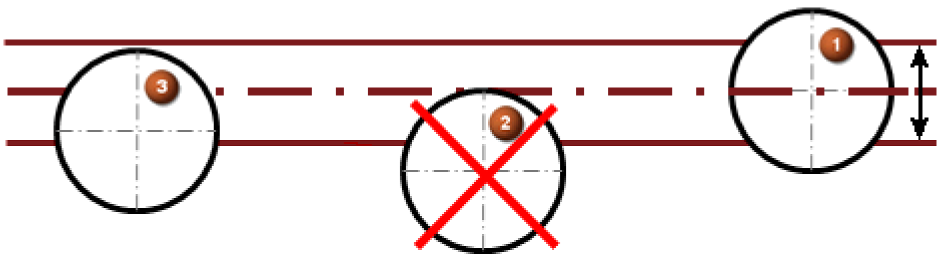
Band Width = 40 mm

The first hole in each pass determines the center line for the pass. NX calculates the band width from this center line, and the band width determines which holes to include in the pass. If you reverse the machining order, a hole that was previously included in the pass could be excluded. NX machines only those holes whose center lines fall within the band width.

In this example, the center lines of holes 2 and 3 are within the band width for the pass. NX includes holes 1, 2, and 3 in the pass.



When the order is reversed, the center line of hole 2 is not within the band width for the pass. NX includes only holes 1 and 3 in the pass.



Note

You must enter the **Band Width** value each time you optimize the sequence.

The options in the **Sequence** group in the **Hole or Boss Geometry** and **Feature Geometry** dialog boxes always display the default settings. The sequencing options are *actions* that NX executes when you click **Reorder List**. NX saves the resulting sequence without saving the settings.

Where do I find it?

Initial release	NX 9.0.3 MP2
Application	Manufacturing
Prerequisite	Hole machining sequence with the **Optimization** option set to **Primary Direction**.
Location in dialog box	**Hole or Boss Geometry** or **Feature Geometry** dialog box→**Sequence** group→**Band Width**

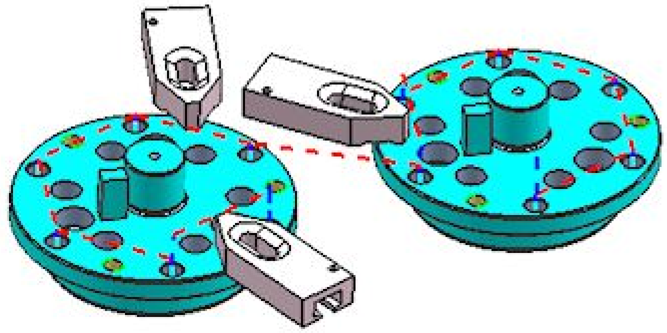
Reduce transition motions and machining time for drilling programs (10.0.1)

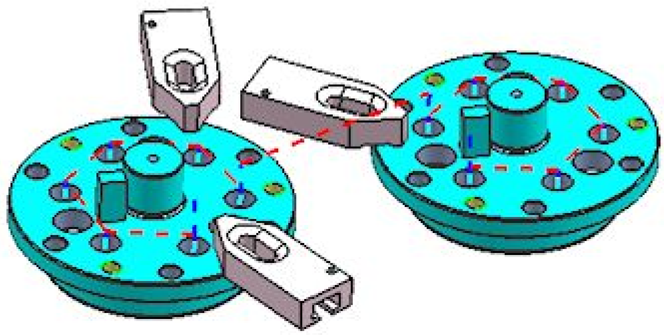
You can now optimize drilling operations across multiple features and workpieces to minimize transition motions and machining time within a program.

- Reorder operations to minimize tool changes.
- Create optimization groups for further optimization and verification.
- Apply operation level sequencing at the optimized group level.
- Generate and verify tool paths at the optimized group level. The **Optimize NC** dialog box includes the standard operation tool path actions.

Before optimizing

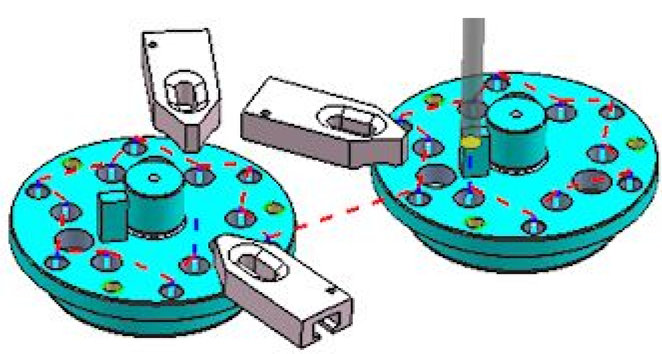
Operation Navigator, **Geometry** view	**Operation Navigator**, **Program Order** view
WORKPIECE **FG_STEP1HOLE** **SPOT_DRILLING** **DRILLING_UP** **DRILLING_FINISH** **FG_STEP1HOLE_STEP1POCKET** **SPOT_DRILLING_2** **DRILLING_UP_2**	**PROGRAM** **SPOT_DRILL** Contains two operations **SPOT_DRILLING** **SPOT_DRILLING_2** **DRILL** **DRILLING_UP** **DRILLING_FINISH** **DRILLING_UP_2**





After optimizing

Operation Navigator, **Geometry** view	**Operation Navigator**, **Program Order** view
WORKPIECE **FG_STEP1HOLE** **SPOT_DRILLING** **DRILLING_UP** **DRILLING_FINISH** **FG_STEP1HOLE_STEP1POCKET** **SPOT_DRILLING_2** **DRILLING_UP_2**	**PROGRAM** **SPOT_DRILL** **OPTIMIZED_NC** **SPOT_DRILLING** **SPOT_DRILLING_2** **DRILL** **DRILLING_UP** **DRILLING_FINISH** **DRILLING_UP_2**



Where do I find it?

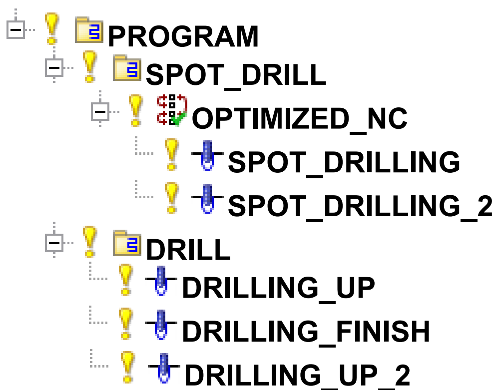
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Operation Navigator	Right-click the program→**Object**→**Optimize**

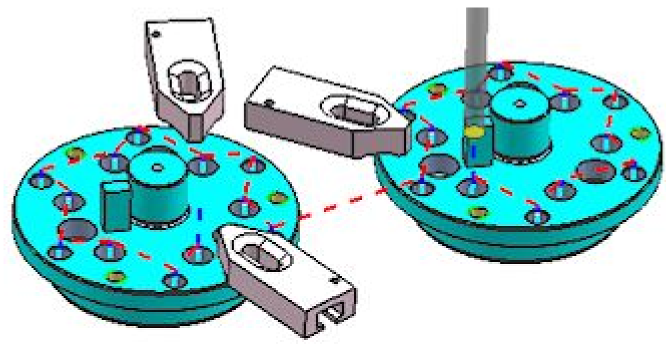
Controlling non cutting moves for an optimized program group (10.0.1)

When you optimize multiple drilling operations at the program level, NX creates an optimized program group, **OPTIMIZED_NC**. You can now specify non cutting moves for this program group, and these moves override the non cutting moves defined in the individual operations.

By default, NX uses the non cutting moves defined in the first operation for the optimized program group. You can use the default settings, or change the non cutting move settings for this program group.

Operation Navigator, **Program Order** view





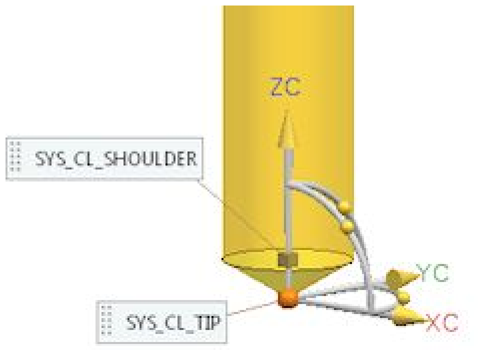
Where do I find it?

Initial release	NX 10.0.1
Application	Manufacturing
Operation Navigator	Right-click the program→**Object**→**Optimize**
Location in dialog box	**Optimize NC** dialog box→**Non Cutting Moves**

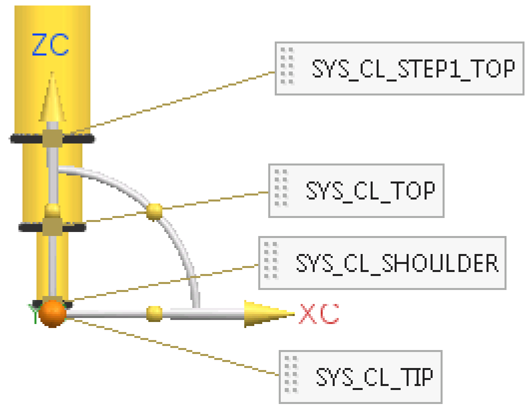
Tracking points for drilling tools

A tracking point is the internal reference point on the tool that NX uses to calculate tool paths.

Drilling tools have two system-defined tracking points at the center line, **SYS_CL_SHOULDER** and **SYS_CL_Tip**.



Step drills have an additional system-defined tracking point at the center line of each step.



For example, **SYS_CL_STEP1_TOP** is located at the top of the first step.

You can also define additional tracking points anywhere on the tool to use in your operation.

Using the tracking points in the drilling operation

You can specify separate tracking points for the non cutting and the cutting moves of a drilling operation.

- To ensure that the tool tip clears the part, select the tracking point at the tool tip for non cutting moves.
- To ensure that a position on the tool reaches a certain depth, select the tracking point that corresponds to that position. For example, select the shoulder of a drill, or a specific shoulder of a step drill.

NX automatically changes to the correct tracking point for each type of move. For example, the drilling operation could use the following system-defined tracking points:

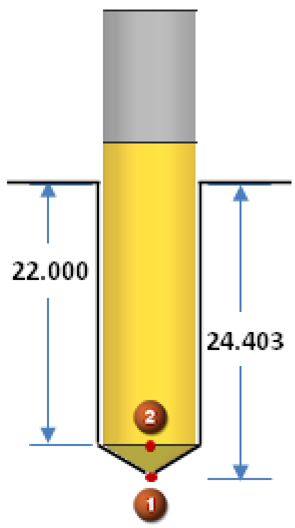
- **SYS_CL_Tip** at the tool tip for all positioning and traversal moves.
- **SYS_CL_SHOULDER** at the tool shoulder for the cutting moves within the drilling cycle.

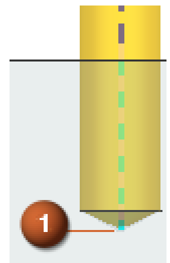
If the cycle output contains rapid moves, such as those for Break Chip or for interrupted holes, the drilling operation would then use:

- **SYS_CL_Tip** for traversal, approach, and departure moves.

- **SYS_CL_SHOULDER** for engage, cutting, and retract moves within the drilling cycle.

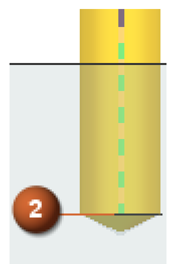
Example outputs for different drilling cycle tracking points





1 **SYS_CL_TIP** output:

```
GOTO/-25.352,89.572,-24.403
```



2 **SYS_CL_SHOULDER** output:

```
GOTO/-25.352,89.572,-22.000
```

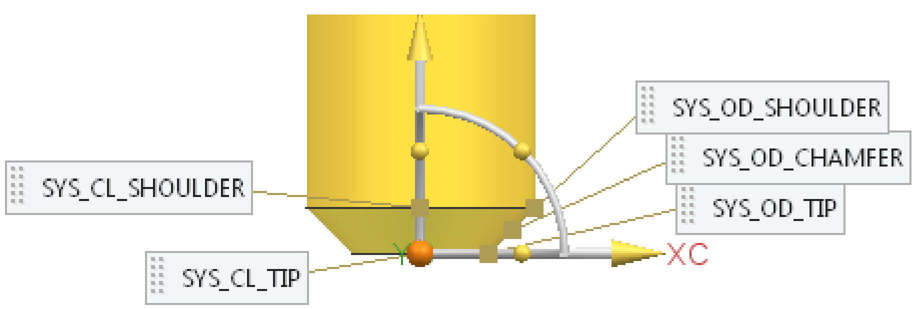
Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	Drilling tool
Location in dialog box	Define the tracking point: Drilling tool dialog box→**More** tab→**Tracking** group→**Tracking Points**→**Tracking Points** dialog box Set the tracking point for non cutting positioning moves: [Drilling operation dialog box]→**Tool** group→**Cutter Compensation** subgroup→**Tracking Data** list **Note** To select a specific tracking point, you must customize the drilling operation dialog box to include the cutter compensation **Tracking Data** list. By default, NX uses the **None** option, which outputs the tool path at the tool tip centerline. Set the tracking point for the drilling cycle: [Drilling operation dialog box]→**Path Settings** group→**Cycle Tracking Data** list

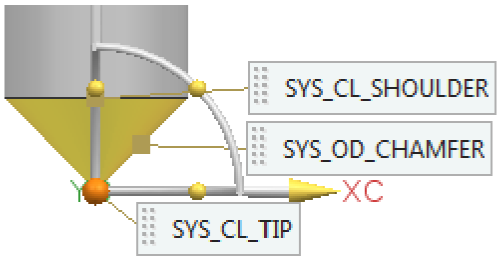
Tracking points for chamfering tools (10.0.1)

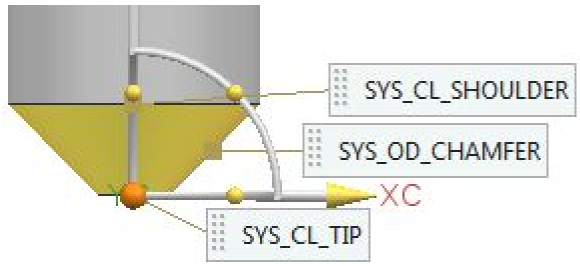
A tracking point is the internal reference point on the tool that NX uses to calculate tool paths.

Chamfer milling tools have two system-defined tracking points at the center line and three system-defined tracking points at the shoulder.



Spot drills and countersinking tools have an added tracking point in the middle of the chamfer.



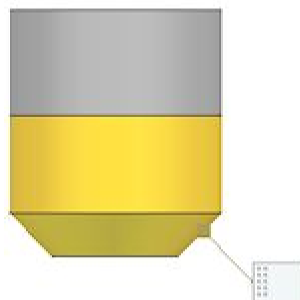


You can also define additional tracking points anywhere on the tool to use in your operation.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Chamfer milling tool
Location in dialog box	Chamfer milling tool dialog box→**More** tab→**Tracking** group→**Tracking Points**→**Tracking Points** dialog box

Specifying tracking points for chamfered tools



You can specify the tracking point on a chamfered tool without specifying the distance from the tip. To do this, use the **By Diameter** option. NX automatically calculates the correct distance from the tool tip.

If you enter a value that is greater than the maximum tool diameter, NX displays an alert and sets the value to the maximum tool diameter.

To manually specify both the diameter and the distance from the tool tip, use the **Full** option.

Where do I find it?

Initial release	NX 10.0
Application	Manufacturing
Prerequisite	Chamfered tool: chamfer mill, spot drill, or countersink tool
Location in dialog box	Chamfered tool dialog box→**More** tab→**Tracking** group→**Tracking Points** →**Tracking Points** dialog box→**Definition** list→**By Diameter** or **Full**

Controlling retract moves for canned drilling cycles (10.0.1)

You can choose how NX outputs the retract moves for the canned cycles in a drilling operation. To do this, use the **Retract Output Mode** option. Select the setting that matches the output to your controller requirements.

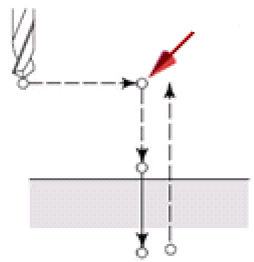
- **Clearance Only** outputs the retract level as part of the canned cycle only when the retract level is equal to the clearance level. In all other cases, NX turns off the cycle and generates an explicit move to the calculated retract point.

- **Clearance Initial** outputs the retract level as part of the canned cycle when the retract level is equal to the clearance level or equal to the initial level.

- **Always** outputs the retract level as part of the canned cycle for postprocessors that can handle this input. The cycle includes the final retract.

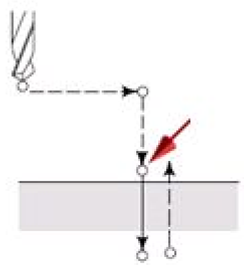
When the tool reaches the bottom of a hole, NX returns the tool to one of the following positions:

1. The initial level, which is the G98 position.



   The initial level is defined as the last Z position before NX begins the cycle.

2. The defined R-plane, which is the G99 position.



   The R-plane is the plane at which the drilling operation begins. NX defines the R-plane and the clearance plane relative to the cycle level. For a constant clearance, if the cycle level changes, so does the R-plane.

   Note

   The initial level does not change when drilling in G99 mode.

3. A Z value on or above the R-plane that is based on the noncutting moves settings.

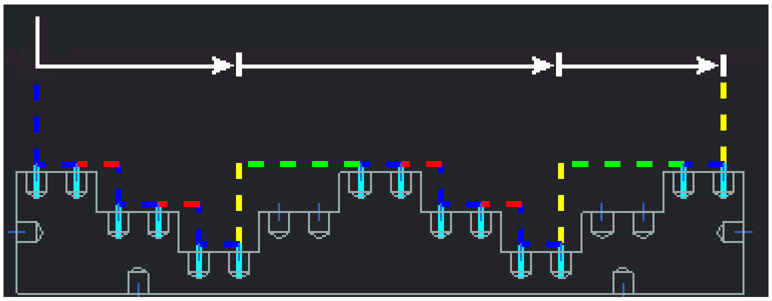
The following examples show how the **Retract Output Mode** option affects the tool paths when using the **Lowest Safe Z**, **Clearance – Tool Axis**, and **Direct** options for transferring between holes. The holes are machined from left to right, omitting holes 7, 8, 15, and 16.

- Yellow = Departure
- Blue = Approach
- Green = Explicit traverse outside of the cycle
- Red = Implicit traverse within the cycle

Lowest Safe Z

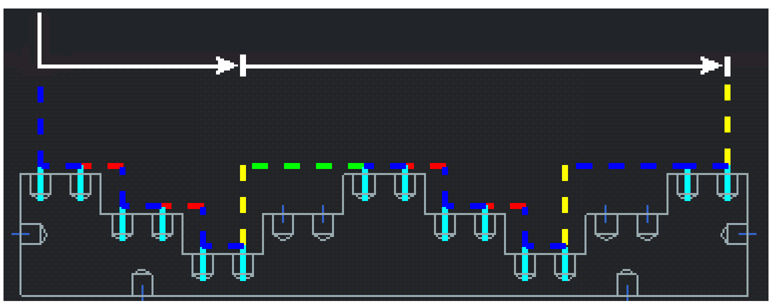
For the **Lowest Safe Z** examples, at the 6th hole, the retract level is below the clearance level, and also below the initial level. At the 14th hole, the retract level is below the clearance level, but at the initial level.

For the **Clearance Only** option, NX outputs a `CYCLE/OFF` statement at the 6th, 14th, and final holes.



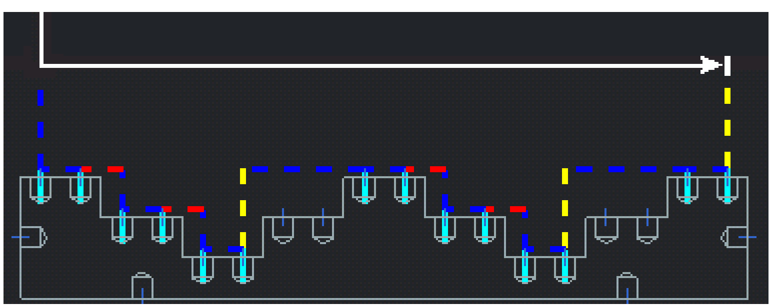
Clearance Only, 3 cycles

For the **Clearance Initial** option, NX outputs a `CYCLE/OFF` statement at the 6th and final holes.



Clearance Initial, 2 cycles

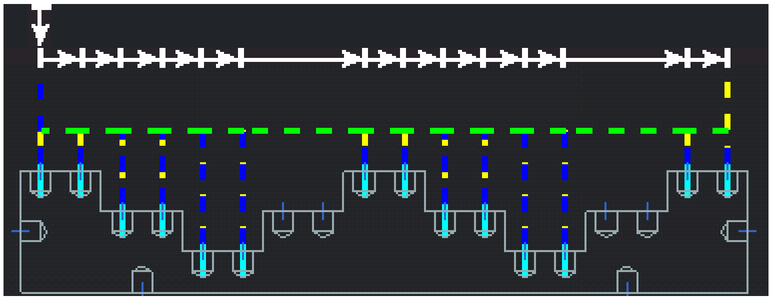
For the **Always** option, NX outputs a `CYCLE/OFF` statement at the final hole.



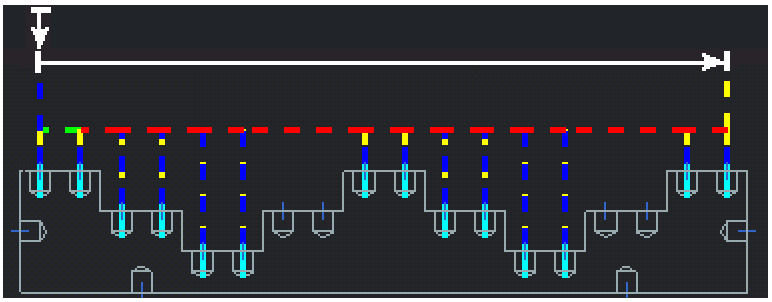
Always, 1 cycle

Clearance – Tool Axis

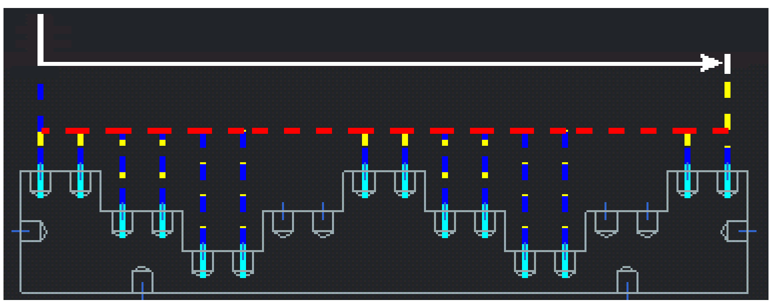
For these examples, the clearance is to the bounding box.



Clearance Only, 14 cycles

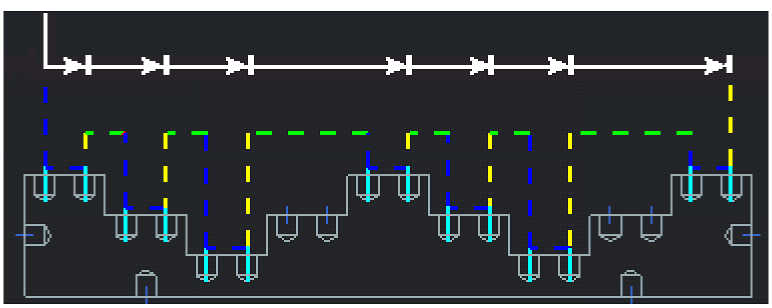


Clearance Initial, 2 cycles

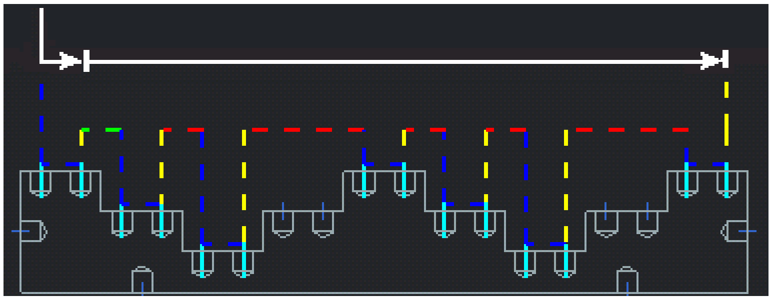


Always, 1cycle

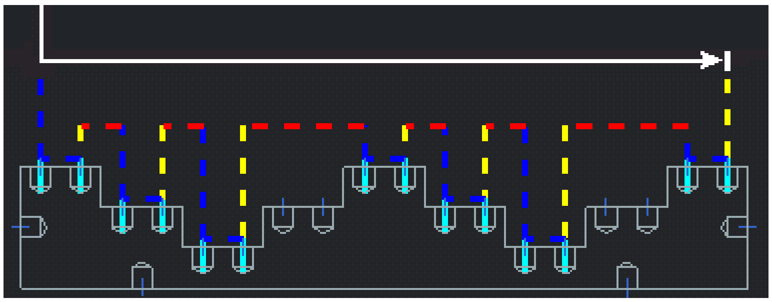
Direct



Clearance Only, 7 cycles



Clearance Initial, 2 cycles



Always, 1 cycle

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Operation **Type** = **hole_making**
Location in dialog box	[Drilling operation] dialog box→**Machine Control** group→**Retract Output Mode** list

Feature group enhancements (10.0.1)

You now have the same control when using an automatic feature group that you have in a manually defined **HOLE_BOSS_GEOM** geometry group. Operations within the feature group geometry parent hierarchy also use the sequence defined in the feature group. In the **Feature Group** dialog box, you can:

- Optimize the sequence of features within the group automatically, using one of the **Optimization** options.
- Reorder the sequence of features within the group manually.

  In the list, select one or more features and use the arrows to move the features as required.
- Reverse the machining direction of selected features.

When this dialog box is open, you can change the orientation of a single feature using the dynamic manipulator.

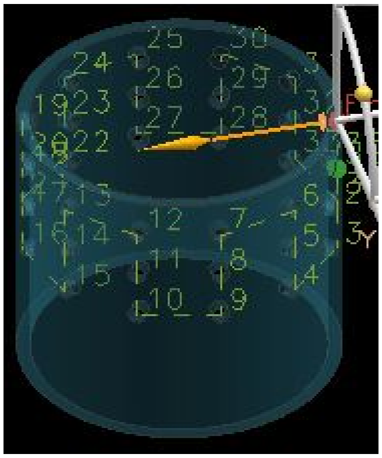
You can control the preview color of the feature sequence using the **Feature Sequence** customer default.

Tip

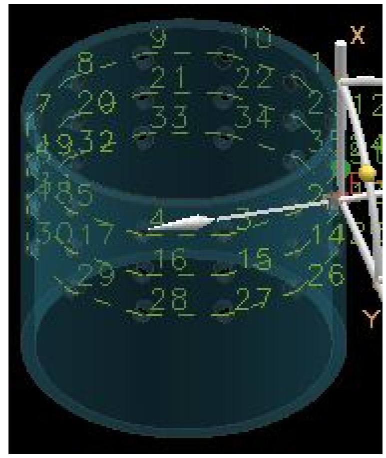
To find a customer default, choose **File**→**Utilities**→**Customer Defaults**, and click **Find Default** .

If needed, you can still access the legacy classification dialog box.

Optimize the feature sequence

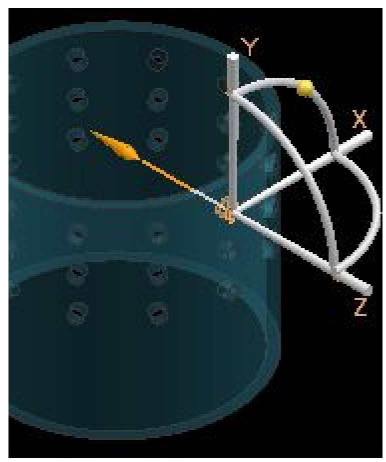


Closest



Primary Direction = X

Reverse machining direction

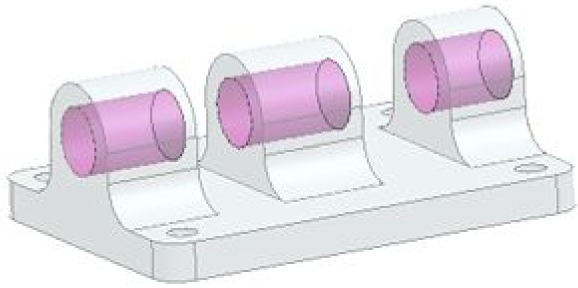


Where do I find it?

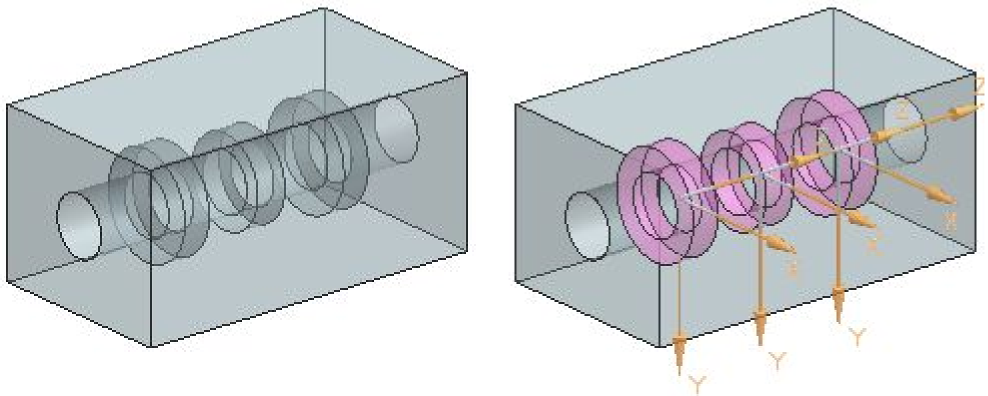
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Operation Navigator	Double-click the feature group

Feature recognition enhancements (10.0.1)

NX now recognizes coaxial interrupted holes. The feature type is **HOLE_ROUND_INTERRUPTED_STRAIGHT**.



NX now recognizes radial grooves. The feature type is **GROOVE_INS_RAD_RECT**.



Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Machining Feature Navigator	Right-click in the background→**Find Features**

Machining unmodeled threads with gouge checking

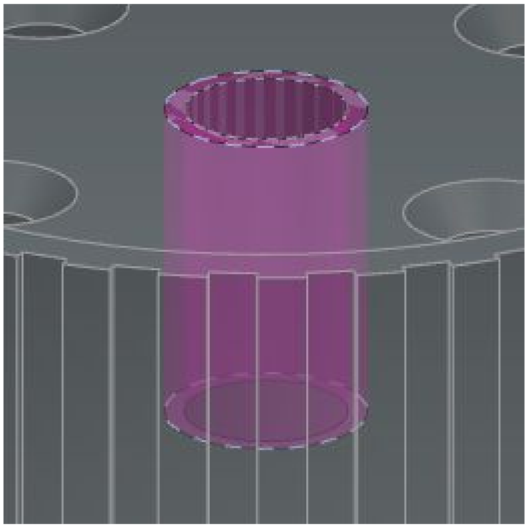
When a modeled hole is the tap drill size, you now can obtain a tool path for tapping operations without turning off gouge checking.

When you generate a tapping operation with the **Gouge Checking** option turned on, NX does not detect a gouge in the threaded volume. Other gouges are still detected and reported.

The **Gouge Check** command and the gouge checking options in verification and simulation no longer report a problem as long as:

- The hole in the part is larger than the minor diameter of the specified thread feature.
- The tool diameter is less than the major diameter of the thread feature.

The threaded volume that is used for gouge checking is the in-process feature for the hole, as shown.



After a gouge check of the tapping operation in the part shown, the **Information** window contains the following output:

```
Operation TAPPING: No gouged motions found.
Hybrid Gouge Checker used.
```

Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Prerequisite	Feature volumes must be defined for the operation.
Operation Navigator	Right-click operation→**Gouge Check** Right-click operation→**Tool Path**→**Verify**→**Replay**→**Gouge and Collision Settings**
Operation dialog box	**Generate** **Verify**→**Replay**→**Gouge and Collision Settings**
Tool Path Editor dialog box	**Analyze Tool Path** group→**Gouge and Collision Check**
Simulation Control Panel dialog box	**Simulation Settings** group→**Simulation Settings** →**Simulation Settings** dialog box→**Collision Detection** group→**Specify Collision Pairs** →**Specify Collision Pairs** dialog box→specify the part and the tool

Allowed violations of part geometry (10.0.1)

In the path settings for an operation, when you check for gouges, NX normally reports a gouge and does not generate a gouging tool path.

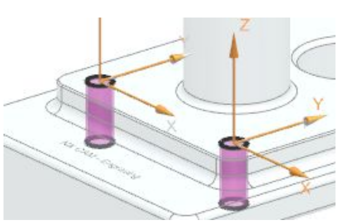
For some situations, you implicitly authorize a gouge to the modeled body by defining a machining feature that is not explicitly modeled. In these situations, NX generates a tool path and does not report gouges within a defined feature volume. For authorized gouges, NX gouge checking reports `No gouged motions found`.

The following are examples of implicitly authorized gouges.

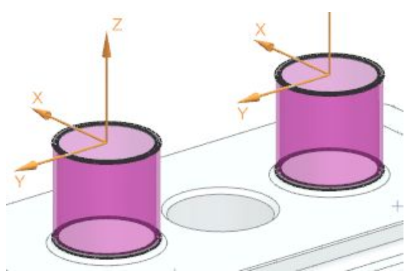
Tapping and thread milling

Typically, you model only the tap diameter of a hole or the maximum thread diameter of a boss. NX creates the feature volume during the machining feature recognition process.

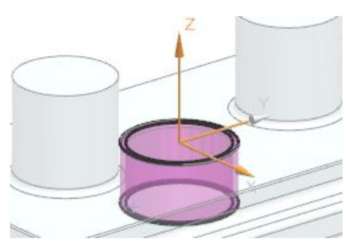
Tapped hole volumes



Thread mill boss volume



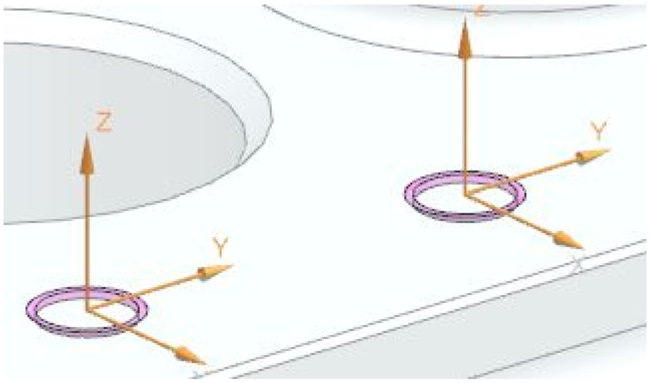
Thread mill hole volume



Chamfering holes

You need not model the chamfer.

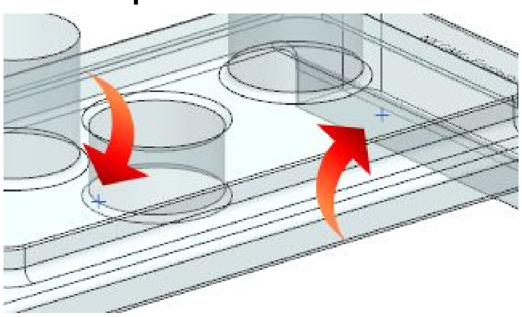
Chamfer volumes



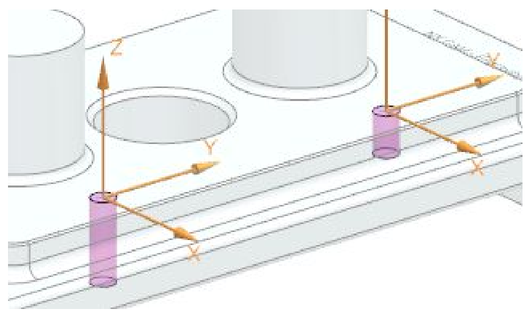
Drilling holes at points

You can specify a hole feature at a point, without modeling the hole.

Modeled points



Hole feature volumes

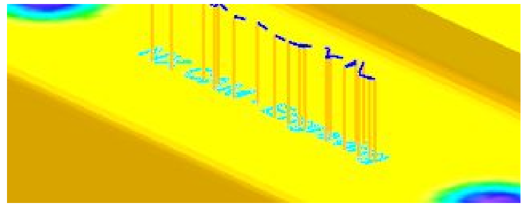


Engraving

When you engrave text using **Planar Text** and **Contour Text** operations, the text to be cut exists only as a 2D note. There is no corresponding feature volume, and the operation dialog box does not contain a gouge checking option. No gouge is reported when you run a gouge check.

2D drafting note

Tool path



Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Feature volumes must be defined for the operation.
Operation Navigator	Right-click operation→**Gouge Check** Right-click operation→**Tool Path**→**Verify**→**Replay**→**Gouge and Collision Settings**
Operation dialog box	**Generate** **Verify**→**Replay**→**Gouge and Collision Settings**
Tool Path Editor dialog box	**Analyze Tool Path** group→**Gouge and Collision Check**
Simulation Control Panel dialog box	For tapping and thread milling only: **Simulation Settings** group→**Simulation Settings** →**Simulation Settings** dialog box→**Collision Detection** group→**Specify Collision Pairs** →**Specify Collision Pairs** dialog box→specify the part and the tool

Integrated Simulation and Verification - ISV

Finding machine, tool, and device components in a CAM or CMM setup

What is it?

In native NX, when you open a part that contains a CAM setup, NX Manufacturing has to find the component parts for machine tools that you added for simulation. In previous releases, when the assembly load options **Load** list was set to the default **From Folder**, NX searched for component parts in the folder specified by the following environment variable:

```
UGII_CAM_LIBRARY_INSTALLED_MACHINES_DIR
```

Beginning in NX 10, NX extends the search to the library tools and devices folders specified by the following variables:

- `UGII_CAM_LIBRARY_TOOL_DIR`
- `UGII_CAM_LIBRARY_DEVICE_DIR`

In CMM inspection, the search is extended to the folders specified by the following variables:

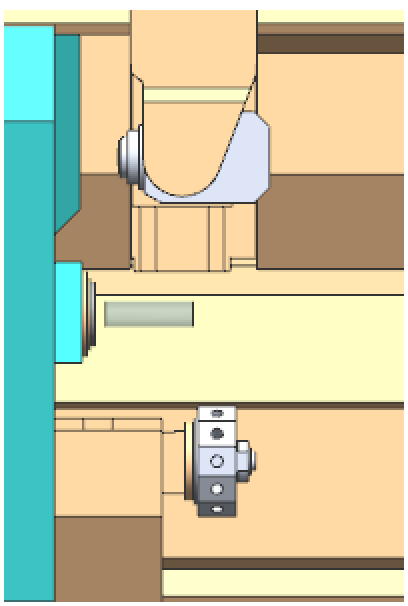
- `UGII_INSPECTION_LIBRARY_INSTALLED_MACHINES_DIR`
- `UGII_INSPECTION_LIBRARY_TOOL_DIR`
- `UGII_INSPECTION_LIBRARY_DEVICE_DIR`

Why should I use it?

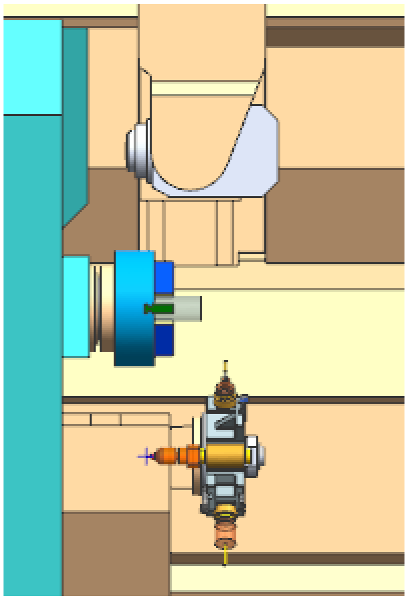
You can accept the default assembly load option **Load** list setting, **From Folder**, even when your CAM setup includes machine tools, solid cutting tools, and devices.

Note

If you modify the standard locations, you must correctly set the variables shown.



In previous releases, if you did not set search folders, NX did not display devices such as chucks and the tailstock, and the cutting tools, in the *sim15_millturn_setup_sinumerik_mm_dual_sync.prt* sample setup assembly.



In NX 10, although you initially get a warning that several components are not found, NX Manufacturing locates the tools and devices and displays them.

Where do I find it?

Initial release	NX 10.0
Application	Manufacturing
Prerequisite	Open a part that has a CAM setup

Saving and retrieving simulation settings

Use the **Save Simulation Settings** and **Load Simulation Settings** commands to save and load a settings file that sets the default simulation parameters in the following dialog boxes:

- **Simulation Settings** and **Animation** groups in the **Simulation Control Panel** dialog box
- **Simulation Settings** dialog box
- **Collision Pairs** dialog box

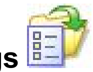
You may have different simulation tasks, such as fast and rough checks, or long detailed checks. You can save time by saving common simulation parameters in different settings files and loading the one that you want when you need it. The files are saved to and retrieved from the folder specified by the **Saved Simulation Settings Directory** simulation customer default.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

The file name of the current settings file is shown in the **Current Settings** row of the **Manage Settings** group. If you do not specify a settings file, NX displays the label **None**.

Where do I find it?

Initial release	NX 9.0.1
Application	Manufacturing
Location in dialog box	**Simulation Control Panel** dialog box→**Simulation Settings** group→**Manage Settings** subgroup→**Load Simulation Settings**  and **Save Simulation Settings** 

Axis dynamic capabilities in the Machine Tool Navigator (10.0.1)

What is it?

Dynamic properties of axes are now stored in a common location, the NX machine tool assembly. You enter the properties from the **Machine Tool Navigator**, using the **Edit**→**Axis** command. The **Machine Tool Navigator** is available in the Manufacturing and Machine Tool Builder applications.

The following properties are new in NX.

For rotary axes and spindles	**For linear axes**
Max Acceleration [deg/s^2]	**Max Acceleration [mm/s^2]**
Max Deceleration [deg/s^2]	**Max Deceleration [mm/s^2]**
Jerk Limit [deg/s^3]	**Jerk Limit [mm/s^3]**
Jump Velocity [deg/s]	**Jump Velocity [mm/s]**
Kv	**Kv**
Fine Precision [deg]	**Fine Precision [mm]**
Coarse Precision [deg]	**Coarse Precision [mm]**

Units of mm for linear axes are shown. For English units, **in** replaces **mm**.

When you retrieve a machine tool model, NX checks whether machine tool axis and channel data has ever been modified or imported, and notifies you if the data has not been updated.

You can import axis dynamic and channel data from an MCF file or enter it manually. You will receive a one-time message that the link between the kinematic model and the MCF is broken, and that simulation will use the axis dynamic data in the part file.

Why should I use it?

- You do not have to enter the same information in multiple applications.

- You can enjoy the relative ease of using the Machine Tool Builder application, without having to learn the more advanced **Machine Configurator** dialog box in the common simulation environment (CSE).

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing Machine Tool Builder
Prerequisite	An open NX assembly that contains a machine tool kinematic structure
Machine Tool Navigator	Right-click the root node→**Import Axis and Channel Data** Right-click an axis node→**Edit**→**Axis**→**Dynamic Properties** group

Channels settings moved (10.0.1)

What is it?

You can now assign axes to channels in the **Machine Tool Navigator** in NX.

You can still assign a device or a tool holder to a channel to a in the **Edit Machine Component** dialog box; however, the options to rename, add, or delete a channel are consolidated in the **Channel Configuration** dialog box.

Why should I use it?

- You no longer have to use the **Machine Configurator** dialog box of the common simulation environment (CSE) to assign axes to channels.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing Machine Tool Builder
Prerequisite	An open NX assembly that contains a machine tool kinematic structure
Machine Tool Navigator	Right-click the root node → **Channel Configuration**

Toolpath verification and analysis consolidation (10.0.1)

What is it?

In previous releases, new functionality was added beside existing functionality. This resulted in some duplication and confusion over which command to use for a given task. Some older functions are now hidden by default in the **Tool Path Verification** dialog box.

2D Dynamic tab

The **2D Dynamic** tab is now hidden. This protects you from issues that might arise if, for example, a collision occurs but is not visible in the selected 2D view.

To restore the **2D Dynamic** tab, select the **Show 2D Dynamic Page** customer default.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

3D Dynamic tab

To ensure maximum reliability, on the **3D Dynamic** tab, the **Facetted Solid** options are hidden.

We recommend that you click **Show Thickness by Color** to check the amount of excess material that remains on the part.

We recommend that you use the following methods for gouge checking:

- Right-click an operation in the **Operation Navigator** and choose **Tool Path**→**Gouge Check**.
- In the **Tool Path Verification** dialog box, on the **Replay** tab, click **Gouge and Collision Settings**.

To restore the **Facetted Solid** options, select the **Enable Facetted Solid for Gouges and Excess** customer default.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing

Postprocessing

MoriAPT CLSF

NX can now output the CLSF for single channel Mori Seiki machine tools in the MoriAPT format developed by Mori Seiki. DMG/Mori Seiki provides a controller-based postprocessor and simulation package for their CNC machines.

When you output your machining data in the MoriAPT format, you can cut parts on Mori Seiki machines without having to create a postprocessor or postprocess the data in NX.

The MoriAPT format is an extension to the ISO4343 APT language format.

You must use the **CLSF Output** command, choose the **MORIAPT** format, choose a name and location for the file, and create the output by clicking **OK** or **Apply**.

Note

Because there is no NC code file or postprocessor, you can simulate only the tool path in NX.

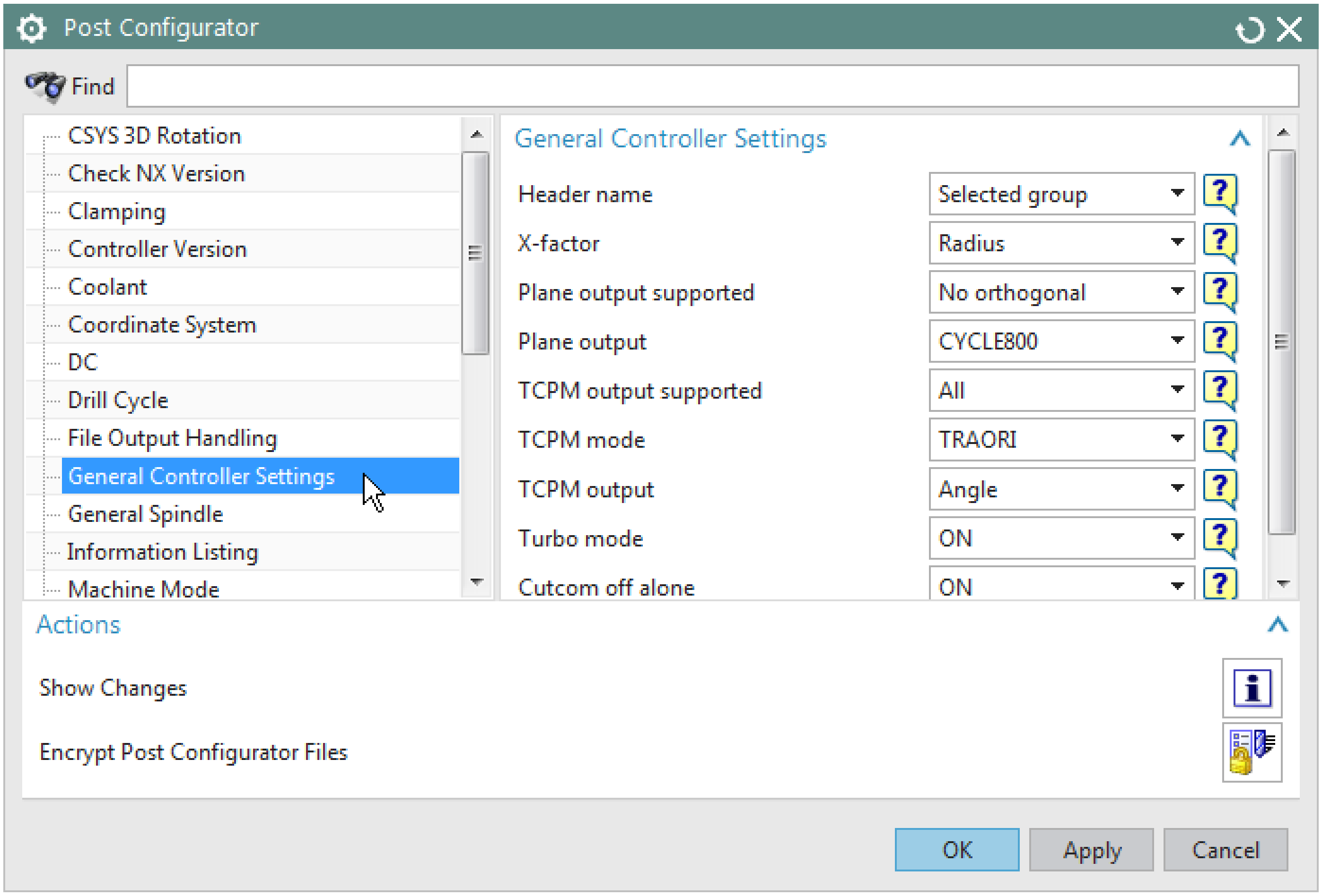
Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Menu	**Menu**→**Tools**→**Operation Navigator**→**Output**→**CLSF**
Ribbon bar	[Select Operation or program group]→**Home** tab→**Operation** group→**More**→**Output CLSF**
Location in dialog box	**CLSF Format** list→**MORIAPT**

Post Configurator (10.0.1)

Use the **Post Configurator** to create postprocessors for the following controllers in an efficient way:

- Sinumerik 840D
- Fanuc 18i
- Heidenhain iTNC 530
- Generic



You can create a new post processor that is based on a loaded kinematic machine model. If the CAM setup does not contain a kinematic model, you must set it up manually.

Why should I use it?

The **Post Configurator** offers a new way of creating postprocessors based on the established manufacturing output manager (MOM) architecture. If you create post processors, you can use and configure the included post libraries. There are three stages of licenses available for NX CAM users.

Basic

This license is available for all NX CAM users. It allows you to:

- Run a postprocessor that was created with Post Configurator.
- Edit parameters to which access is granted by the post processor developer.

Advanced

The license is intended for machine tool manufacturers and field service technicians.

It is available for all NX CAM users who currently own a Post Builder license. In addition to do the tasks that you can do with the basic license you can:

- Create a new post processor.

- Edit your post processor using the Post Configurator dialog box.

Full

The post developer license.

The license must be purchased. It is designed for Siemens Professional Services staff, Siemens partners, and NX CAM users who are heavily involved in creating post processors. In addition to the tasks that you can do with the advanced license, you can also:

- Edit the Tcl code.
- Encrypt your post processors.
- Limit the running of a post processor to a specific Sold-To ID.
- Define an expiration date for your post processors.

Note

When you create post processors using the **Post Configurator**, you can edit them only in the **Post Configurator**. If you created post processors using the **Post Builder**, you will not be able to edit them in the **Post Configurator**.

Where do I find it?

Application	Manufacturing
Ribbon	**Home** tab→**Tools** drop-down→**Post Configurator** 
Menu	**Menu**→**Tools**→**Post Configurator**

Feature-based Machining

Machining through holes and blind holes together

You can group through holes and blind holes to machine them together. You save on programming time because you do not have to program the through holes and blind holes separately. Use the **Predefined** option of the **Group Features** command to group features that have identical diameters, thread pitch, and number of steps. When it groups the features, NX ignores the values of all depth attributes, including tolerances, chamfers and thread lengths.

How it works

In NX, through holes are **STEPxHOLE** features and blind holes are **STEPxPOCKET** features, where x indicates the number of steps in the feature. When you group features using the **Predefined** option, NX:

- Includes all **STEP1HOLE** and **STEP1POCKET** features with identical diameters in an **FG_STEP1HOLE_STEP1POCKET** feature group.

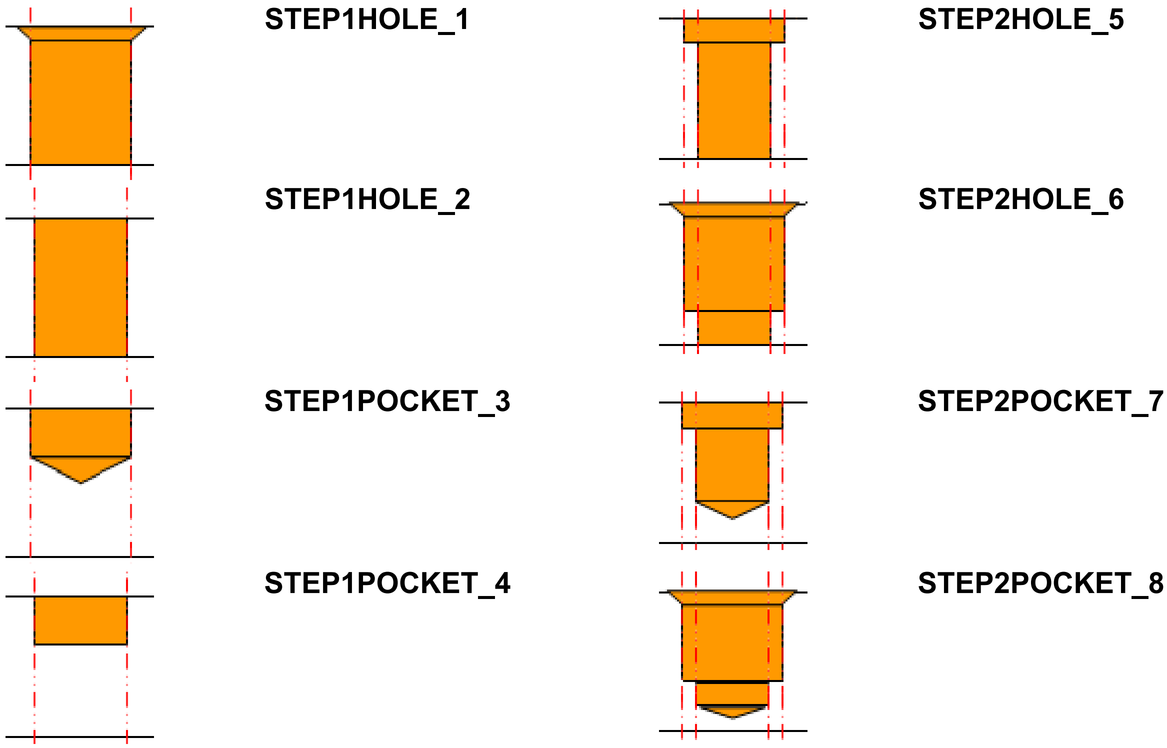
- Includes all **STEP1HOLE_THREAD** and **STEP1POCKET_THREAD** features with identical diameters and thread pitch in an **FG_STEP1HOLE_THREAD_STEP1POCKET_THREAD** feature group.

- Repeats the process for features with multiple steps.

If additional feature groups are required for holes with different diameters, NX creates the feature groups and names them **FG_STEP1HOLE_STEP1POCKET_1**, **FG_STEP1HOLE_STEP1POCKET_2** and so on.

The following example shows different hole feature profiles, and how NX groups the hole features together.

Features to group

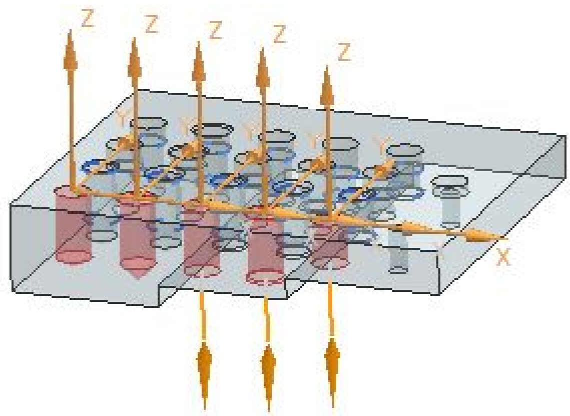


Features in the Machining Feature Navigator after grouping

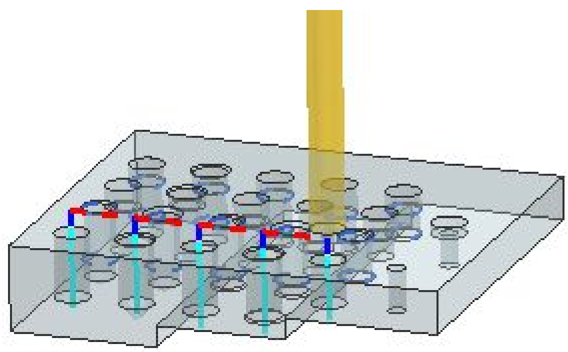
- WORKPIECE
  - FG_STEP1HOLE_STEP1POCKET
    - STEP1HOLE_1
    - STEP1HOLE_2
    - STEP1POCKET_3
    - STEP1POCKET_4
  - FG_STEP2HOLE_STEP2POCKET
    - STEP2HOLE_5
    - STEP2HOLE_6
    - STEP2POCKET_7
    - STEP2POCKET_8

Sample part

Features selected for **FG_STEP1HOLE_STEP1POCKET**



Drilling operation for **FG_STEP1HOLE_STEP1POCKET**



The **Group Features** command also has the following options:

- **None** groups features whether or not the attributes are identical.
- **All** groups features only if all of the attributes are identical.

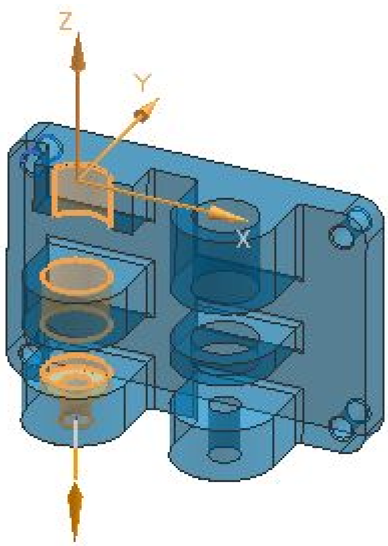
Where do I find it?

Initial release	NX 9.0.2
Application	Manufacturing
Machining Feature Navigator	Right-click in the background→**Group Features** Grouping selected features Right-click selected features→**Group Features**
Location in dialog box	**Group by** group→**Identical Attributes** list →**Predefined**

Feature recognition for coaxial interrupted holes (10.0.1)

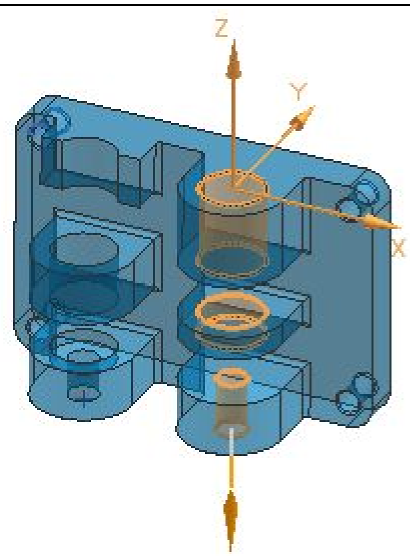
NX first tries to recognize a sequence of coaxial interrupted holes as the appropriate **STEPx** feature so that the machining process is the same for holes with and without interruptions. If NX does not recognize a sequence of coaxial interrupted holes as a regular stepped feature, it classifies the feature as **HOLE_ROUND_INTERRUPTED_STRAIGHT**. The machining process is different for these holes.

In the following examples, both features consist of coaxial cylinders that extend through multiple flanges.



STEP2HOLE

The lowest flange contains a **STEP2HOLE** feature. The additional cylinders that extend through the remaining flanges have the same diameter as the largest **STEP2HOLE** diameter.



HOLE_ROUND_INTERRUPTED_STRAIGHT

The lowest flange contains a **STEP1HOLE** feature. In this case, the diameters of the additional cylinders in the remaining flanges do not match the diameter of the **STEP1HOLE** feature. This hole requires a different machining process than a **STEP1HOLE** feature.

You can use both **STEPx** and **HOLE_ROUND_INTERRUPTED_STRAIGHT** features as input for the **Sequential Drilling** operation.

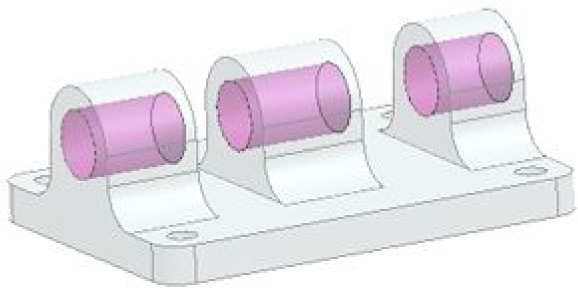
Where do I find it?

Initial release	NX 10.0.1
Application	Manufacturing

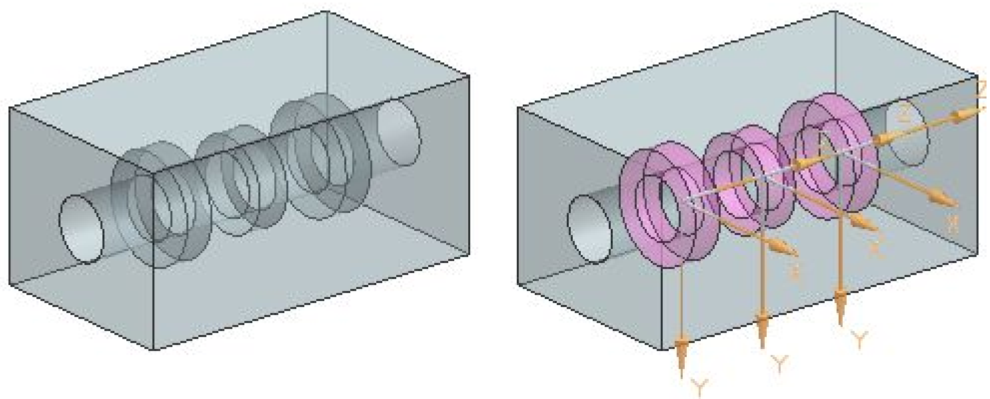
Machining Feature Navigator	Right-click in the background→**Find Features**

Feature recognition enhancements (10.0.1)

NX now recognizes coaxial interrupted holes. The feature type is **HOLE_ROUND_INTERRUPTED_STRAIGHT**.



NX now recognizes radial grooves. The feature type is **GROOVE_INS_RAD_RECT**.



Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Machining Feature Navigator	Right-click in the background→**Find Features**

Chapter 5: CAE

Advanced Simulation

Solver version support (10.0.1)

For each released version of NX, the following tables list the supported solver versions for import, export, and the post-processing of results. Note:

- The version listed in the **Import ASCII** and **Import Binary** rows is the solver version that was generally available when the NX version was released. In general, the import of the solver ASCII and binary files should be upwards compatible. Therefore, you should be able to import them into the most recent version of NX. However, in general:
  - o ASCII files are backwards compatible for import into NX. If you import an ASCII file from a newer version of the solver than is officially supported, the software simply ignores any new fields/options that aren't supported in the current NX release.
  - o Binary files are not backwards compatible. For example, you can import a binary file created by NX Nastran 5.0 into NX 6.0.2, but you might not be able to import a binary file created by NX Nastran 6.1 into NX 5.
- The version listed in the **Export ASCII** rows is the solver version that was available when the NX version was tested. In general, the exported solver input file is upwards compatible for that solver. Backwards compatibility is not guaranteed. For NX Nastran, the **Model Setup Check** function in Advanced Simulation tries to flag potential version incompatibility issues.
- The version listed in the **Post-processing Results** rows is the version of the solver results that was tested in the listed NX version. In general, results from earlier solver versions are also supported.

NX 10 releases

Solver	File Type	NX 10	NX 10.0.1
NX Nastran	Import ASCII (.dat)	10	10
	Import Binary (.op2)	10	10
	Export ASCII (.dat)	10	10
	Post-processing of Results (.op2, .xdb)	10	10.1
MSC Nastran	Import ASCII (.dat)	2013.1	2014
	Import Binary (.op2)	2013.1	2014
	Export ASCII (.dat)	2013.1	2014
	Post-processing of Results (.op2, .xdb)	2013.1	2014

Solver	File Type	NX 10	NX 10.0.1
Abaqus	Import ASCII (.inp)	6.13	6.13
	Import Binary	N/A	N/A
	Export ASCII (.inp)	6.13	6.13
	Post-processing of Results (.fil)	6.14	6.14-1
	Post-processing of Results (.odb)	6.13-5	6.13-5
ANSYS	Import ASCII (PREP7, CDB)	15.0	15.0
	Import Binary (.rst, .rth)	15.0	15.0
	Export ASCII (.inp)	15.0	15.0
	Post-processing of Results	15.0	15.0
LS-DYNA	Import ASCII	971R6.0	971R6.0
	Import Binary	N/A	N/A
	Export ASCII (.k)	971R6.0	971R6.0
	Post-processing of Results	971R7.1	971R7.1
Permas	Post-processing of Results (.res, .post)	13	13
SAMCEF	Import ASCII (.dat)	16.1 (build 16.1-02)	16.1 (build 16.1-02)
	Import Binary (.sdb)	16.1 (build 16.1-02)	16.1 (build 16.1-02)
	Export ASCII (.dat)	16.1 (build 16.1-02)	16.1 (build 16.1-02)
	Post-processing of Results (.des, .fac)	16.1 (build 16.1-02)	16.1 (build 16.1-02)

NX 9 releases

Solver	File Type	NX 9	NX 9.0.1	NX 9.0.2	NX 9.03
NX Nastran	Import ASCII (.dat)	9	9	10	10
	Import Binary (.op2)	9	9	10	10
	Export ASCII (.dat)	9	9	10	10
	Post-processing of Results (.op2)	9	9	10	10
MSC Nastran	Import ASCII (.dat)	2013	2013	2013.1	2013.1
	Import Binary (.op2)	2013	2013	2013.1	2013.1
	Export ASCII (.dat)	2013	2013	2013.1	2013.1
	Post-processing of Results (.op2)	2013	2013	2013.1	2013.1
Abaqus	Import ASCII (.inp)	6.12	6.12	6.12	6.12
	Import Binary	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.12	6.12	6.12	6.12
	Post-processing of Results (.fil)	6.12-1	6.12-1	6.13-4	6.13-4
	Post-processing of Results (.odb)	6.12	6.12	6.12	6.13-2

Solver	File Type	NX 9	NX 9.0.1	NX 9.0.2	NX 9.03
ANSYS	Import ASCII (PREP7, CDB)	14.5	14.5	15.0	15.0
	Import Binary (.rst, .rth)	14.5	14.5	15.0	15.0
	Export ASCII (.inp)	14.5	14.5	15.0	15.0
	Post-processing of Results	14.5	14.5	15.0	15.0
LS-DYNA	Import ASCII	971R6.0	971R6.0	971R6.0	971R6.0
	Import Binary	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R6.0	971R6.0	971R6.0	971R6.0
	Post-processing of Results	971R6.0	971R6.0	971R7.1	971R7.1
Permas	Post-processing of Results (.res)	13	13	13	13

NX 8 releases

Solver	File Type	NX 8	NX 8.0.1	NX 8.0.2	NX 8.0.3	NX 8.5	NX 8.5.1	NX 8.5.2
NX Nastran	Import ASCII (.dat)	8	8	8.5	8.5	8.5	8.5	8.5
	Import Binary (.op2)	8	8	8.5	8.5	8.5	8.5	8.5
	Export ASCII (.dat)	8	8	8.5	8.5	8.5	8.5	8.5
	Post-processing of Results (.op2)	8	8.1	8.5	8.5	8.5	8.5	9
MSC Nastran	Import ASCII (.dat)	2011.1	2011.1	2012.1	2012.1	2012.1	2012.1	2012.1
	Import Binary (.op2)	2011.1	2011.1	2012.1	2012.1	2012.1	2012.1	2012.1
	Export ASCII (.dat)	2011.1	2011.1	2012.1	2012.1	2012.1	2012.1	2012.1
	Post-processing of Results (.op2)	2011.1	2011.1	2012.1	2012.1	2012.1	2012.1	2013
Abaqus	Import ASCII (.inp)	6.10	6.10	6.10	6.10	6.12	6.12	6.12
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.10	6.10	6.10	6.10	6.12	6.12	6.12
	Post-processing of Results (.fil)	6.11	6.11	6.11	6.12-1	6.12-1	6.12-1	6.12-1
	Post-processing of Results (.odb)	6.10-EF1	6.11	6.11	6.11	6.12	6.12	6.12

Solver	File Type	NX 8	NX 8.0.1	NX 8.0.2	NX 8.0.3	NX 8.5	NX 8.5.1	NX 8.5.2
ANSYS	Import ASCII (PREP7, CDB)	13	13	14	14	14	14	14.5
	Import Binary (.rst, .rth)	13	13	14	14	14	14	14.5
	Export ASCII (.inp)	13	13	14	14	14	14	14
	Post-processing of Results	13	13	14	14	14	14	14
LS-DYNA	Import ASCII	971R5.0	971R5.0	971R5.0	971R5.0	971R6.0	971R6.0	971R6.0
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R5.0	971R5.0	971R5.0	971R5.0	971R6.0	971R6.0	971R6.0
	Post-processing of Results	971R5.0	971R5.0	971R5.0	971R5.0	971R6.0	971R6.0	971R6.0

NX7 releases

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2	NX 7.5.3	NX 7.5.4	NX 7.5.5.
NX Nastran	Import ASCII (.dat)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Import Binary (.op2)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Export ASCII (.dat)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Post-processing of Results	6.1	7.0	7.1	7.1	7.1	7.1	8
MSC Nastran	Import ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Import Binary (.op2)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Export ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Post-processing of Results	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
Abaqus	Import ASCII (.inp)	6.8-1	6.9–1	6.9–1	6.9-1	6.10	6.10	6.10
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9	6.9	6.9	6.10	6.10	6.10
	Post-processing of Results (.fil)	6.8-EF2	6.9.2	6.9.2	6.10-1	6.10-1	6.10-1	6.11-1
	Post-processing of Results (.odb)	6.8-EF2	6.9-EF1	6.9-EF2	6.9-EF2	6.10-EF1	6.10-EF1	6.10-EF1

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2	NX 7.5.3	NX 7.5.4	NX 7.5.5.
ANSYS	Import ASCII (PREP7, CDB)	12	12.1	12.1	12.1	13	13	13
	Import Binary (.rst, .rth)	12	12.1	12.1	12.1	13	13	13
	Export ASCII (.inp)	12	12.1	12.1	12.1	13	13	13
	Post-processing of Results	12	12.1	12.1	12.1	12.1	12.1	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1

NX 6 releases

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3	NX 6.0.4	NX 6.0.5
NX Nastran	Import ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Import Binary (.op2)	6.0	6.1	6.1	6.1	6.1	7.0
	Export ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Post-processing of Results	6.0	6.0	6.1	6.1	7.0	7.0
MSC Nastran	Import ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Import Binary (.op2)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Post-processing of Results	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
Abaqus	Import ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Post-processing of Results (.fil)	6.7-5	6.8-1	6.8-3	6.8-EF2	6.8-EF2	6.8-EF2
	Post-processing of Results (.odb)	N/A	N/A	N/A	6.8-EF	6.8-EF2	6.9-EF2

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3	NX 6.0.4	NX 6.0.5
ANSYS	Import ASCII (PREP7, CDB)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Import Binary (.rst, .rth)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Export ASCII (.inp)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Post-processing of Results	11 SP1	11 SP1	11 SP1	11 SP1	12.0	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R2	971R2	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A	N/A	N/A	N/A	N/A

NX 5 releases

Solver	File Type	NX 5	NX 5.0.1	NX 5.0.2	NX 5.0.3	NX 5.0.4	NX 5.0.5	NX 5.0.6
NX Nastran	Import ASCII (.dat)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Import Binary (.op2)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Export ASCII (.dat)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Post-processing of Results	5.0	5.0	5.1	5.1	5.1	5.1	6.0
MSC Nastran	Import ASCII (.dat)	2005	2005	2007	2007	2007	2007	2007r1
	Import Binary (.op2)	2005	2005	2007	2007	2007	2007	2007r1
	Export ASCII (.dat)	2005	2005	2007	2007	2007	2007	2007r1
	Post-processing of Results	2005	2005	2007	2007	2007	2007	2008r1
Abaqus	Import ASCII (.inp)	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.7-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.7-1
	Post-processing of Results	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.8-1
ANSYS	Import ASCII (PREP7, CDB)	10	10	11	11	11	11	11
	Import Binary (.rst, .rth)	10	10	11	11	11	11	11
	Export ASCII (.inp)	10	10	11	11	11	11	11
	Post-processing of Results	10	11	11	11	11	11	11 SP1

NX 4 releases

Solver	File Type	NX 4	NX 4.0.1	NX 4.0.2	NX 4.0.3	NX 4.0.4
NX Nastran	Import ASCII (.dat)	4.0	4.1	4.1	5.0	5.0
	Import Binary (.op2)	4.0	4.1	4.1	4.1	4.1
	Export ASCII (.dat)	4.0	4.1	4.1	5.0	5.0
	Post-processing of Results	4.0	4.1	4.1	5.0	5.0
MSC Nastran	Import ASCII (.dat)	2005	2005	2005	2005	2005
	Import Binary (.op2)	2005	2005	2005	2005	2005
	Export ASCII (.dat)	2005	2005	2005	2005	2005
	Post-processing of Results	2005	2005	2005	2005	2005
Abaqus	Import ASCII (.inp)	6.5-1	6.5-1	6.5-1	6.6	6.6
	Import Binary	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.5-1	6.5-1	6.5-1	6.6	6.6
	Post-processing of Results	6.5-1	6.5-1	6.5-1	6.6	6.6-3
ANSYS	Import ASCII (PREP7, CDB)	8	9	9	10	10
	Import Binary (.rst, .rth)	8	9	9	10	10
	Export ASCII (.inp)	8	9	9	10	10
	Post-processing of Results	9	9	9	10	10

General capabilities

Measuring the area and perimeter of polygon faces

What is it?

You can now use the **Measure Face** command to calculate the area and perimeter values of polygon body faces. You can also perform the calculation in different analysis units. If you select the **Associative** check box, NX creates:

- A **Face Measure** feature in the history for the FEM file.
- An expression for the area and the perimeter of each polygon face.

Note

If you clear the **Associative** check box, NX does not create the **Face Measure** feature or the expressions.

Additionally, you can now use the **Measure Area** option in the **Expressions** dialog box to calculate the area and perimeter of polygon body faces. With this option, NX always creates the **Face Measure** feature and the expressions for area and perimeter for each polygon face.

In previous releases, you could use **Measure Face** or **Measure Area** to calculate the area and perimeter of body faces only in the CAD part.

Where do I find it?

Measure Face

Application	Advanced Simulation
Prerequisite	An active FEM file
Command Finder	**Measure Face** 
Menu	**Analysis→Measure Face**

Measure Area

Application	Advanced Simulation
Prerequisite	An active FEM file
Menu	**Tools→Expressions**
Location in dialog box	**Measurements** list

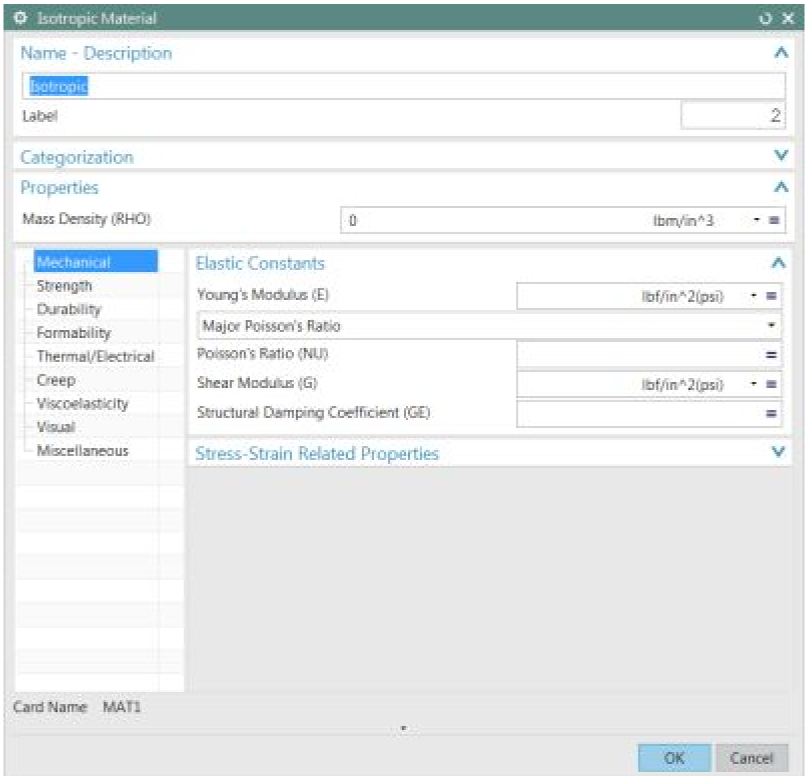
Enhancements to dialog boxes with tabs

What is it?

Dialog boxes that had multiple horizontal tabs in previous releases of NX have been enhanced to a vertical layout. In previous releases, NX may have run slowly when you used these dialog boxes. The new vertical layout is also easier to use.

The following are examples of dialog boxes that have been updated to this layout:

- **Solution** dialog box
- **Solution Step** dialog box (NX Multiphysics environment only)
- **Materials** dialog boxes
- **Output Requests** dialog box



Where do I find it?

Application	Advanced Simulation

Alternative workflow for editing 3D mesh and element properties

What is it?

You can now edit mesh and element properties for a 3D mesh using the **Mesh Properties** command. In previous releases, the command supported only 2D meshes.

The **Mesh Properties** command lets you specify all mesh and element properties for a selected mesh. This command is an alternative to the standard workflow of defining these properties using individual commands such as **Physical Properties**, **Edit Mesh Collector**, and **Mesh Associated Data**.

Note

This command affects the properties only for the selected mesh, and not any other meshes or elements. This is different from the standard workflow, in which you can share and inherit properties across many meshes.

The **Mesh Properties** dialog box has three tabs:

- **Element Properties** — Lets you define element properties such as material orientation. This tab corresponds to the **Mesh Associated Data** dialog box.

- **Collector Properties** — Lets you change the type of mesh collector in which this mesh is stored. This tab corresponds to the **Mesh Collector** properties dialog box.

- **Physical Properties** — Lets you define physical properties such as default thickness, nonstructural mass, and material properties. This tab corresponds to the **Physical Property Table** dialog box.

You can define the same properties for several selected meshes at the same time. To do this, select the meshes and then open the **Mesh Properties** dialog box.

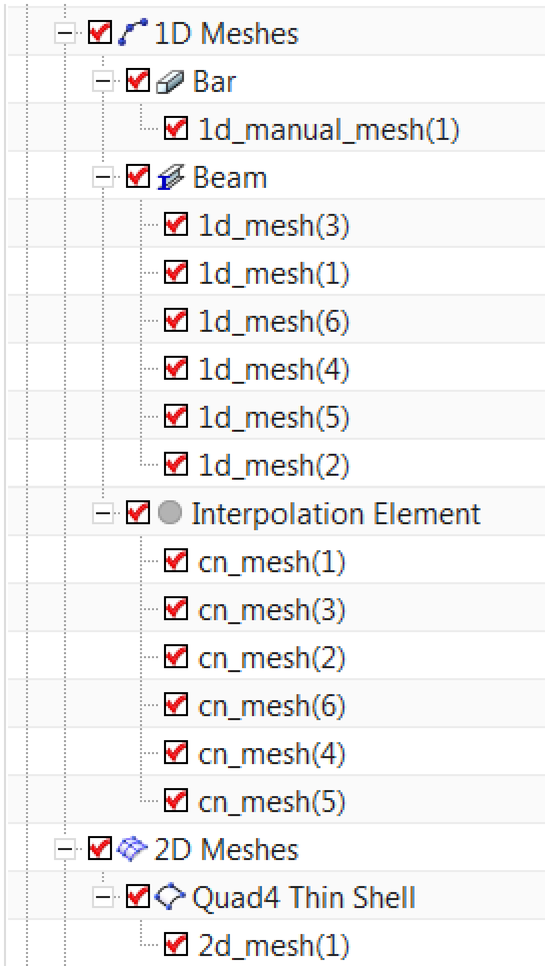
Where do I find it?

Application	Advanced Simulation
Prerequisites	A FEM file as the work part and displayed part The **Simulation Navigator** set to **Mesh View**
Command Finder	**Mesh Properties** 
Graphics window	Right-click one or more 2D or 3D meshes → **Mesh Properties**
Simulation Navigator	Right-click one or more 2D or 3D meshes → **Edit Mesh Properties**

Additional grouping in Mesh View of Simulation Navigator

What is it?

When you view your meshes in the **Mesh View** of the **Simulation Navigator**, meshes are now grouped according to their mesh type in addition to their element family.



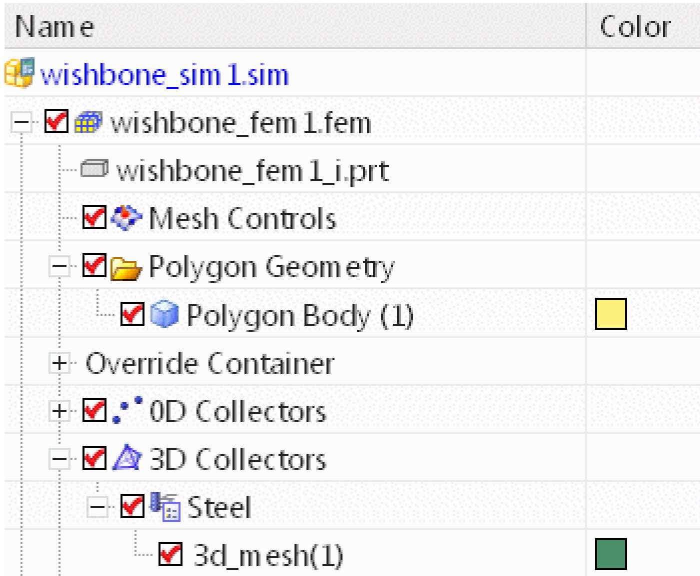
In previous versions, the **Mesh View** grouped the meshes only by element family.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the **Simulation Navigator** column heading and choose **Mesh View**.

Identifying objects by color

In the **Simulation Navigator**, the **Color** column lets you view and modify colors of various entities in the graphics window.



You can view and modify colors for the following entities:

- Polygon bodies
- Meshes
- Coordinate systems
- Boundary conditions

  These include loads, constraints, and Simulation objects.

- Physical properties

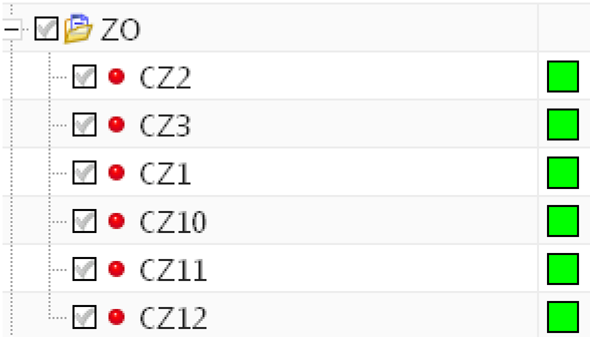
  The **Color** column for physical properties is visible only in the **Physical Properties** view. To make the colors visible in the **Physical Properties** view, in the **Model Display** dialog box, on the **Element** tab, set **Color Basis** to **Physical Property Table**.

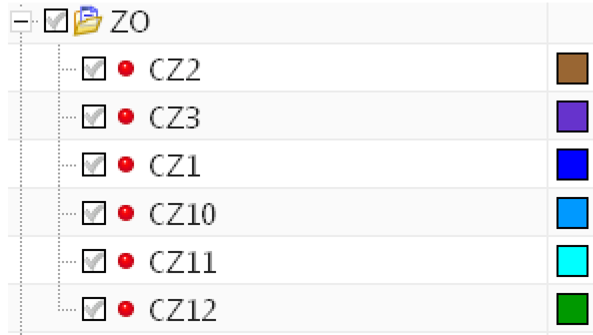
- Materials

  The **Color** column for materials is visible only in the **Material** view. To make the colors visible in the **Material** view, in the **Model Display** dialog box, on the **Element** tab, set **Color Basis** to **Material Property Table**.

To modify the color of an object, double-click the color in the **Color** column.

To assign a series of different colors to multiple boundary conditions, select the boundary conditions or a folder containing multiple boundary conditions. Right-click and select **Sequence Display Colors**.





Original colors

Colors after using Sequence Display Colors

To modify the colors in the sequence, in the **Customer Defaults** dialog box, choose **Simulation→Boundary Conditions**, and click the **Sequence Display Colors** tab.

Where do I find it?

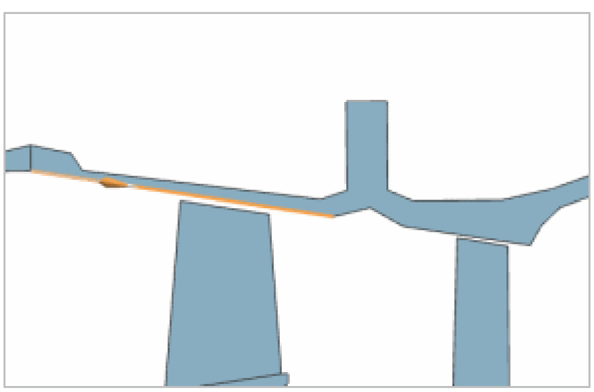
Application	Advanced Simulation, Design Simulation
Prerequisite	Boundary conditions applied to the model
Simulation Navigator	Right-click multiple boundary conditions→**Sequence Display Colors**

Edge selection enhancements

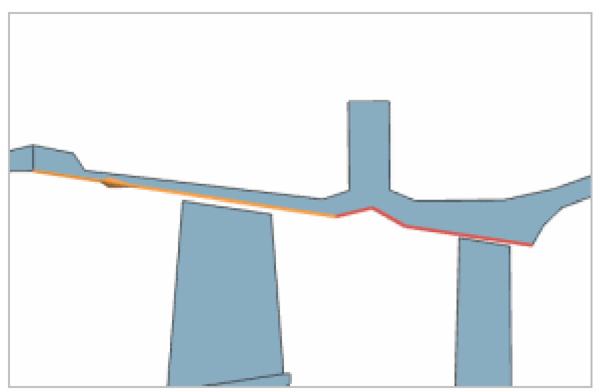
What is it?

This release includes the following new methods to select connected edges.

- The **Connected Edges** method helps you select connected polygon edges.

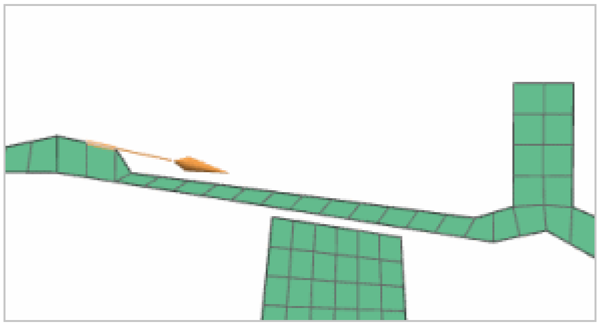


First edge selected

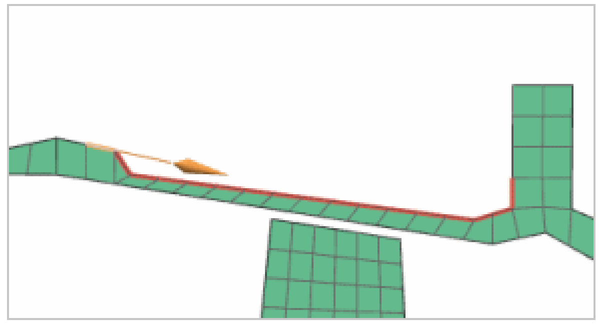


Connected edges highlighted

- The **Connected Element Edges** method helps you select connected element edges.



First element edge selected



Connected element edges highlighted

To start, select one or more polygon or element edges. When you move your mouse, NX highlights additional connected polygon or element edges that you can select. Use the **Smart Selector Options** dialog box to enter additional selection criteria for connected edges. You can specify free edges, gaps, and gap tolerance.

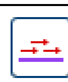
This release also includes enhancements to path selection for the **Thermal Stream** command. You can now select a single path of polygon edges that span multiple polygon bodies that are not topologically connected. In previous releases, you could select only paths on edges that were topologically connected. Use the new **Path Selector Options** dialog box to specify free edges, gaps, and a maximum gap tolerance between edges.

Where do I find it?

Edge selection

Application	Advanced Simulation
Selection bar	**Method** list

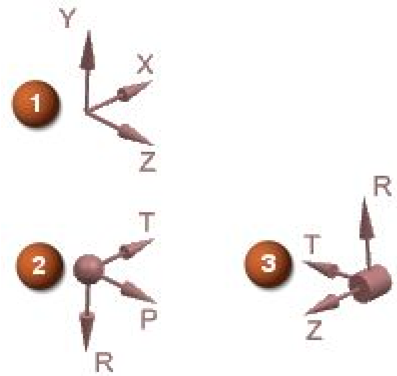
Path selection

Application	Advanced Simulation
Command Finder	**Thermal Stream** 
Simulation Navigator	Right-click the **Load Container** node→**New Load**→**Thermal Stream**

Representative graphics for local coordinate systems

What is it?

In this release, the graphical icons that appear in the NX graphics window for local coordinate systems now visually indicate whether the coordinate system is Cartesian, spherical, or cylindrical.



(1) Cartesian; (2) Spherical; (3) Cylindrical

Also, in the **Visualization Preferences** dialog box, you can now control the size of the coordinate system display and control its visibility through the model. For example, to improve dynamic viewing performance in models that have thousands of coordinate systems, you can turn off the coordinate system display. For more information about new options related to coordinate systems and the View Triad, see CSYS and View Triad enhancements.

Where do I find it?

Application	Gateway, Advanced Simulation
Command Finder	**Visualization Preferences**
Location in dialog box	**Visualization Preferences** dialog box→**View/Screen** tab→**Part Settings** group
Simulation Navigator	Right-click **CSYS**→**Display Settings**

Solution Monitor enhancements

What is it?

- The **Solution Monitor** is now used by the NX Multiphysics and NX Nastran solvers.
- The **Solution Monitor** now displays text output for each solver on a separate tab when you run a coupled NX Multiphysics analysis.
    - The **NX Multiphysics** tab displays the text output for the coupled solver.
    - The **NX Thermal** tab displays the text output for the thermal solver.
    - The **NX Nastran** tab displays the text output for the NX Nastran solver.
- You can copy and paste the displayed text output. In previous versions, the displayed text output was not selectable.
- The user interface of the **Solution Monitor** is refreshed and the command names are more intuitive. The commands now have tooltips describing their functionality.

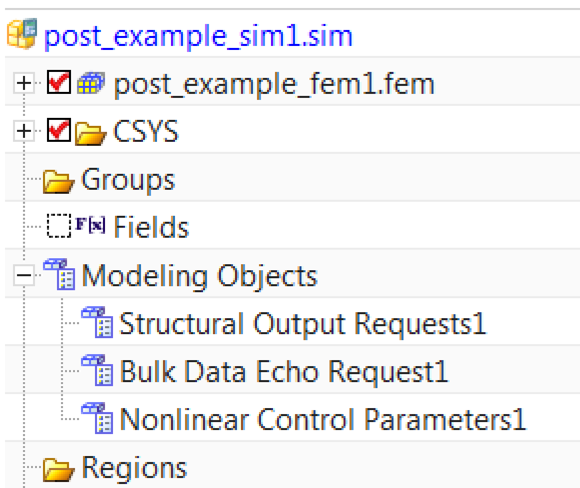
Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	NX Nastran, NX Multiphysics, NX Thermal and Flow, NX Space Systems Thermal, or NX Electronic Systems Cooling as the selected solver environment
Command Finder	**Solve**
Simulation Navigator	Right-click a solution node→**Solve**

Viewing modeling objects in the Simulation Navigator

What is it?

You can view modeling objects in the **Simulation Navigator**. You can sort and filter the modeling objects that will be visible. By default, the filter is set to **All Off**.



Support for non-ASCII characters

What is it?

You can now use multi-byte characters to name files and enter data in the Advanced Simulation, Design Simulation, and Motion applications. For example, you can use multi-byte characters in the names of:

- All Motion objects, such as joints and drivers.
- Solutions, solution steps, or subcases.
- CAE related files, such as FEM and Simulation files.

In NX, your specified locale controls the language used in the user interface as well as the code set for storing character data. For example, if your locale is set to China, you can enter Chinese character data in the NX user interface.

The graphic shows Chinese characters in the **Simulation Navigator**.

NX follows the Unicode international standard to store multi-byte character data, using UTF–8 encoding. Although NX supports the use of multi-byte characters, many third party software packages that interface with NX, such as finite element solvers, support only ASCII character data.

Caution

NX does not provide an error message if the solver receives unsupported characters. You must verify that your solvers will support your desired locale before implementing localization.

Solver support for multi-byte characters

Solver	Unicode support	Solve with non-ASCII entities	Solve with non-ASCII file names	Export with non-ASCII file names and entities	Import with non-ASCII file names and entities
NX Nastran	No	Yes	Yes	Yes	Yes
MSC Nastran	No	No	No	Yes	Yes
Abaqus	No	No	Yes	Yes	Yes
ANSYS	No	No	No	Yes	Yes
NX Nastran Design	No	Yes	Yes	Yes	Yes
NX Multiphysics (Structural)	No	Yes	Yes	N/A	Yes
NX Multiphysics (Thermal/Mapping/Coupled)	No	No	No	N/A	Yes

Solver	Unicode support	Solve with non-ASCII entities	Solve with non-ASCII file names	Export with non-ASCII file names and entities	Import with non-ASCII file names and entities
NX Thermal/Flow (**Thermal**/**Advanced Thermal**,**Flow/Advanced Flow**, **Coupled Thermal-Flow**, **Axisymmetric Thermal**)	No	No	No	No	Yes
NX Thermal/Flow (**Mapping/Axisymmetric Mapping**)	No	Yes	Yes	No	Yes
LS-DYNA	No	Yes	Yes	Yes	Yes
UNV	No	N/A	N/A	Yes	Yes
Shape Optimization	No	No	No	N/A	N/A
Topology Optimization	No	No	No	N/A	N/A
Durability	No	No	Yes	N/A	N/A
Geometry Optimization	No	Yes	Yes	N/A	N/A
Adaptivity	No	Yes	Yes	N/A	N/A
NX Motion Co-Simulation	No	No	No	N/A	N/A
NX Motion Recurdyn	No	Yes	Yes	N/A	N/A
NX Motion ADAMS	No	Yes	Yes	N/A	N/A

Fields

Complex data support

What is it?

When you create a table field, you can now define tabular data that has a complex dependent domain. You can:

- Import the tabular data from a text file.
- Edit or create the tabular data in an external spreadsheet application.

- Enter the tabular data directly.

Just like table fields with real domains, table fields with complex dependent domains are defined from the **Table Field** dialog box. To specify a complex dependent domain, you must first access the unfiltered listing of dependent domains.

The software accepts complex data in the real/imaginary format which is the default, or the magnitude/phase format. To switch to the magnitude/phase format, in the **Table Field** dialog box, in the **Data Points** table, right-click the **(Real)** or **(Imaginary)** column heading and choose **Display Magnitude/Phase**.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click **Fields**→**New Field**→**Table**
Location in dialog box	**Table Field** dialog box→Click **Remove complex domains filter** →**1–D** list→**Other**→Independent domains list→select **ComplexNumber**

Extrapolation options

What is it?

You can now specify the extrapolation method when any Delaunay method is selected as the interpolation method in the **Table Field** dialog box. In earlier versions of NX, you could specify the extrapolation method only when **Linear** was selected as the interpolation method.

When you select a Delaunay method as the interpolation method in the **Table Field** dialog box, you have the following extrapolation options: **Undefined** and **Constant**.

- Select **Undefined** if you do not want the software to assign a value to the dependent domain of points whose independent domain lies outside the cloud of points defined by the tabular data.

- Select **Constant** if you want the software to do the following:

  1. Identify the location on the surface of the cloud of points that is closest to the point whose independent domain lies outside the cloud of points.

  2. Interpolate the tabular data to calculate the value for the dependent domain at that location on the surface of the cloud of points.

  3. Use the interpolated value as the table lookup value for the point whose independent domain lies outside the cloud of points.

The **Constant** option in NX 10 is equivalent to the default behavior when a Delaunay method is selected as the interpolation method in earlier versions of NX.

Where do I find it?

Application	Advanced Simulation

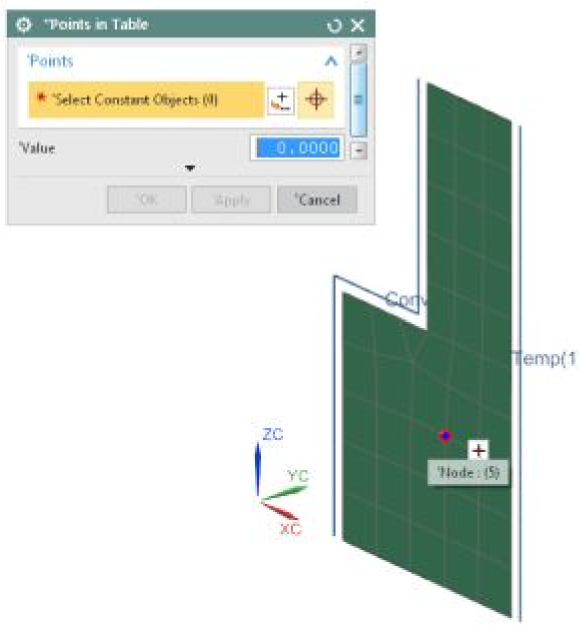
Simulation Navigator	Right-click **Fields**→**New Field**→**Table**
Location in dialog box	**Table Field** dialog box→**Interpolation** list→**Delaunay – Fast** or **Delaunay – Medium** or **Delaunay – Accurate**→**Values Outside Table** list→**Constant** or **Undefined**

Graphical table editing

What is it?

When you create a spatial field, you can now obtain the coordinates for the tabular data directly from the graphics window. In previous versions of NX, you had to explicitly enter the coordinates for the tabular data. After you select a point in the graphics window, you must enter the corresponding value of the dependent domain.

To support the new functionality, the new **Points in Table** dialog box is available. You type the value of the dependent domain for the selected point in the **Value** field in this dialog box.



Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click **Fields**→**New Field**→**Table**
Location in dialog box	**Table Field** dialog box→**Data Points** group→**Graphical table editing** →**Points in Table** dialog box

Creating an axisymmetric spatial field

What is it?

You can now create spatial fields with axisymmetric independent domains. The utility of an axisymmetric spatial field is that NX can apply the axisymmetric spatial field, which is two-dimensional, across a three-dimensional model. To facilitate this capability, **Axisymmetric Plane** is added to the list of independent domains options in the **Formula Field** and **Table Field** dialog boxes.

Typical applications for this capability include:

- Using axisymmetric thermal results in a 3D structural model
- Mapping axisymmetric pressure data to a 3D surface

In both of these applications, you can use either a table field or a formula field.

- If you can describe how the dependent domain varies in the axisymmetric plane with a closed-form mathematical expression, use a formula field.
- If you can describe how the dependent domain varies in the axisymmetric plane only with tabular data, use a table field.

Where do I find it?

Application	Advanced Simulation
Prerequisite	In the **New FEM** or **New FEM and Simulation** dialog box, in the **Solver Environment** group, from the **2D Solid Option** list, select **ZX Plane, Z Axis** or **XY Plane, X Axis**.
Simulation Navigator	Right-click **Fields**→**New Field**→**Expression** or **Table**
Location in dialog box	**Formula Field** or **Table Field** dialog box→**Domain** group→**Independent** list→**Axisymmetric Plane**

4D field support

What is it?

Beginning with NX 10, you can create 4D fields. *4D fields* are formula fields or table fields whose independent domain includes both spatial coordinates and a single non-spatial variable like time, temperature, or frequency. Mathematically, 4D fields allow you to model functional relationships of the form:

$$f = f(x_1,x_2,x_3,\alpha)$$

where x_1, x_2, and x_3 represent a set of Cartesian, cylindrical, spherical, or parametric spatial coordinates, and α represents a non-spatial variable like time, temperature, or frequency.

You can use 4D fields to define the following boundary conditions:

- Acceleration
- Heat flux

- Heat generation
- Nodal pressure
- Pressure
- Radiation
- Temperature load
- Convection
- Enforced displacement
- Thermal constraints

In earlier releases, you could define the magnitude of these boundary conditions only as a function of a single non-spatial variable and define the distribution of these boundary conditions as a function of spatial coordinates. The value of the boundary condition that NX applies at a node is the product of the two functions.

For example, suppose the magnitude is defined as a function of time and the distribution is defined as a function of Cartesian coordinates. The value of the boundary condition applied to node *i* at time t_o is as follows:

$$bc_i = f(t_o)\ g(x_i,y_i,z_i)$$

where:

- bc_i is the boundary condition value for node *i* at time t_o
- $f(t_o)$ is the time-dependent function for the magnitude at time t_o
- $g(x_i,y_i,z_i)$ is the spatial function for the distribution
- (x_i,y_i,z_i) are the Cartesian coordinates for node *i*

You can define the functional relationships using either formula fields or table fields, or both a formula field and a table field.

The 4D field capability allows you to define both magnitude and distribution with a single function. Thus, for the previous example, the value of the boundary condition applied to node *i* at time t_o is:

$$bc_i = h(x_i,y_i,z_i,t_o)$$

You can define the functional relationship using either a formula field or a table field.

If the 4D functional relationship can be expressed as a closed-form mathematical relationship, you can create the 4D field as a formula field. When NX evaluates the formula field for a given set of spatial and non-spatial values, it simply evaluates the expression that relates the independent and dependent domains at the given values. Because the evaluation is direct, formula fields are not subject to interpolation errors.

If the 4D functional relationship can only be approximated by a set of tabular data that relates the independent and dependent domains, you can create the 4D field as a table field. When NX evaluates the table field for a given set of spatial and non-spatial values at which a tabular data point does not exist, it interpolates the tabular data to obtain the corresponding dependent domain value.

You can select the interpolation method that NX uses to look up values. The interpolation methods applicable to 4D table fields are:

- Nearest neighbor and approximate nearest neighbor methods
- Renka's modified Shepard method
- Inverse distance weighting method

These interpolation methods treat the non-spatial variable as a spatial variable in the interpolation calculations. This is true even though the dimensions of the non-spatial variable typically differ from the dimensions of the spatial variables.

If the table look up is only at locations where a data point is defined, the 4D interpolation effectively becomes an interpolation over only the non-spatial variable. For such a case, errors arising from treating the non-spatial variable as a spatial variable do not occur. However, for all other cases, you should first consider the following alternatives:

- If you can relate the independent and dependent domains with a closed-form mathematical relationship, create a 4D formula field instead of a 4D table field.
- If you can group the tabular data into a reasonable number of sets of spatial data points with each set corresponding to a specific value of the non-spatial variable, create a table of fields instead of a 4D table field. With a table of fields, two distinct interpolations occur during a table look up. The first interpolation is over the spatial domain. The second interpolation is over the non-spatial domain.

If a 4D formula field or a table of fields are not a viable alternative, and the table look up is not only at locations where a tabular data point is defined, you can still minimize 4D interpolation errors as follows:

- Define enough tabular data points to ensure that a tabular data point is in close spatial and non-spatial proximity to any look-up point.
- Use interpolation sub-options like **Approximate Nearest Neighbor Tolerance** and **Interpolate On** to limit the distance from the look-up point that a tabular data point can be located and still be included in the interpolation calculation.

Where do I find it?

Application	Advanced Simulation
Command Finder	**Temperature Load**
Location in dialog box	**Magnitude** group→**Temperature** list→**New Field**→**Formula Constructor** or **Table of Fields Constructor**→**Domain** group→**Independent** list→**Time, Cartesian**

Scaling fields that define boundary conditions

What is it?

In earlier versions of NX, when you defined a boundary condition, the **Scale Factor** box appeared in the **Magnitude** group after you selected **Field** from the drop-down list.

In NX 10, the **Scale Factor** box does not appear when you define the boundary condition with a field.

Now, the edit button appears. To enter a scale factor, click , choose **Scale Factor** from the list, and enter your value in the **Scale Factor** box in the **Field Scale Factor** dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Define the magnitude and distribution of a boundary condition with a field
Location in dialog box	Boundary condition dialog box→**Magnitude** group→Edit →list→**Scale Factor**→**Field Scale Factor** dialog box→**Scale Factor** box

Scaling fields that define material properties

What is it?

You can scale fields that define material properties. In earlier versions of NX, when you used a field to define a material property, you were not given an option to scale the field. Now, after you assign a field to a material property, the edit button appears. To enter a scale factor, click , choose **Scale Factor** from the list, and enter your value in the **Scale Factor** box in the **Field Scale Factor** dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Define a material property with a field
Location in dialog box	Material property dialog box→Material property group→Edit →list→**Scale Factor**→**Field Scale Factor** dialog box→**Scale Factor** box

Expressions

Auto-generated expressions

What is it?

Now NX automatically generates expressions for a set of commonly used variables such as Cartesian coordinates, frequency, time, temperature, and so on. For many of these variables, the names of the auto-generated expressions and the variable names are identical. Because you can enter expressions directly in the formulas for other expressions, the need to use the UG variable function is eliminated.

For example, NX auto-generates an expression for the temperature variable. The name of the auto-generated expression is *temperature*.

In earlier versions of NX, to enter the temperature variable in the formula for an expression, you would type:

```
ug_var("temperature")
```

Now, to enter the temperature variable in the formula for an expression, you can type:

```
temperature
```

In this example, `temperature` is an expression name. Because expression names are not string constants, you do not surround them in quotes.

NX also auto-generates expressions for any variables that you use to define the independent and dependent domains of a field that otherwise would not be auto-generated.

For example, suppose you select strain as the independent variable of a field. When you create the field, NX auto-generates an expression named *strain*. To include the strain variable in the formula for an expression, you can type:

```
strain
```

where `strain` is an expression name.

When NX auto-generates expressions, it disallows blank spaces and some characters, and replaces them as follows.

Disallowed character	Disallowed character description	Replacement	Replacement character description
	Blank space	_	Underscore
.	Period	_	Underscore
(	Parenthesis	_	Underscore
)	Parenthesis	None. The character is dropped.	
{	Brace	None. The character produces an error.	
}	Brace	None. The character produces an error.	
&	Ampersand	None. The character produces an error.	
–	Dash	None. The character produces an error.	

All expressions must have unique names. Because NX auto-generates expressions for certain variables, duplicate expression names would occur if you opened a part that contains an expression whose name is identical to one of the auto-generated expression names. To prevent this from occurring, use the **Variable Prefix**, **Rename Existing Expressions**, and **Rename Prefix** customer defaults.

- If the **Rename Existing Expressions** customer default is enabled, NX pre-pends the text string defined by the **Rename Prefix** customer default to any existing expression names that are the same as an auto-generated expression name. NX does not use the **Variable Prefix** customer default when **Rename Existing Expressions** is enabled.

- If the **Rename Existing Expressions** customer default is disabled, NX pre-pends the text string defined by the **Variable Prefix** customer default to any auto-generated expression names that are the same as an existing expression name. NX does not use the **Rename Prefix** customer default when **Rename Existing Expressions** is disabled.

Where do I find it?

Application	Advanced Simulation
Command Finder	**Customer Defaults**
Location in dialog box	**Simulation→Fields→Rename Existing Expressions** or **Rename Prefix** or **Variable Prefix**

Referencing fields in expressions

What is it?

Beginning with NX 10, you can reference a field in the formula for an expression with the new *field function*. The field function is notated as `fd( )`. The argument for the field function is the name of the field that is being referenced and optional entries that control how NX evaluates the field.

For example, you can apply the field function to a field that represents a function of a single independent variable. Mathematically, such a function is expressed as $f = f(x)$, where the independent variable might be a spatial coordinate or a non-spatial variable like time or temperature.

Suppose the **Simulation Navigator** lists a table field named `Table Field(1)` that relates how pressure varies as a function of time. To enter this field in the formula for an expression such that pressure is returned at any time for which the expression is evaluated, in the formula, type:

```
fd(“Table Field(1)”)
```

Note

You must place quotes around string constants. Names of fields are string constants.

To enter this field in the formula for an expression such that the field is evaluated at a specific time, in the formula, type:

```
fd(“Table Field(1)”,t0)
```

where `t0` is the numerical value for the specified time.

You can also apply the field function to a field that represents a multi-variable functional relationship like a function of two independent variables. Mathematically, such a function is expressed as $f = f(x,y)$, where the independent variables might be two spatial coordinates, or a single spatial coordinate and a non-spatial variable like time or temperature, or a pair of non-spatial variables.

Suppose the **Simulation Navigator** lists a formula field named `Formula Field(1)` that relates how pressure varies as a function of X- and Y-coordinates. To enter this field in the formula for an expression such that pressure is returned for any combination of X- and Y-coordinates for which the expression is evaluated, in the formula, type:

```
fd("Formula Field(1)")
```

To enter this field in the formula for an expression such that the field is evaluated at a specific X-coordinate, but over all Y-coordinates, in the formula, type:

```
fd("Formula Field(1)",X0,y)
```

where `X0` is the numerical value for the specified X-coordinate.

Note

The order of independent variables is given in the **Domain** group of the dialog box for the field that represents the multi-variable function.

To enter this field in the formula for an expression such that the field is evaluated at a specific Y-coordinate, but over all X-coordinates, in the formula, type:

```
fd("Formula Field(1)",x,Y0)
```

where `Y0` is the numerical value for the specified Y-coordinate.

To enter this field in the formula for an expression such that the field is evaluated at a specific X-coordinate and a specific Y-coordinate, in the formula, type:

```
fd("Formula Field(1)",X0,Y0)
```

where `X0` and `Y0` are the numerical values for the specified X- and Y-coordinates, respectively.

With the field function, you can also do the following:

- Include mathematical operations within the argument of the field function
- Define composite functions
- Reference components of vector-valued functions

New functions for expressions

What is it?

New functions are introduced in NX 10 for defining expressions. The SIGN(v) function is implemented across all NX applications. The other functions are supported only in the NX CAE application.

New general-purpose functions

Function	Description
DR("point name")	Returns the radial displacement of the input point. The DR("point name") function is results dependent.
DX("point name")	Returns the X-displacement in the global Cartesian coordinate system at the input point. The DX("point name") function is results dependent.

Function	Description
DY(“point name”)	Returns the Y-displacement in the global Cartesian coordinate system at the input point. The DY(“point name”) function is results dependent.
DZ(“point name”)	Returns the Z-displacement in the global Cartesian coordinate system at the input point. The DZ(“point name”) function is results dependent.
FD(“field name”)	Returns the value for a field.
GAPD(“point name”)	Returns the local gap distance at the input point. The GAPD(“point name”) function is results dependent.
KU(v)	Converts temperature to Kelvin and removes the units.
R(“point name”)	Returns the radial distance from the axis of rotation to the input point.
RS(“point name”)	Returns the rotational speed at the input point.
SIGN(v)	Returns -1 if the input value is negative, 0 if the input value is 0, +1 if the input value is positive.
TEMP(“point name”)	Returns the temperature of the solid at the input point.
THICK(“point name”)	Returns the thickness at the input point.
X(“point name”)	Returns the X-coordinate in the global Cartesian coordinate system at the input point.
Y(“point name”)	Returns the Y-coordinate in the global Cartesian coordinate system at the input point.
Z(“point name”)	Returns the Z-coordinate in the global Cartesian coordinate system at the input point.

New fluid-thermal functions

Function	Description
ABS2REL	Returns the relative temperature of a fluid at a given absolute temperature, radius, rotational speed, and fluid swirl velocity.
ABS2REL_SR	Returns the relative temperature of a fluid at a given absolute temperature, radius, rotational speed, and fluid swirl ratio.
CONDFL(v)	Returns the thermal conductivity of a fluid at a given temperature.
DENSFL(v)	Returns the density of a fluid at a given temperature.
ENTHFL(v)	Returns the specific enthalpy of a fluid at a given temperature.
GAMMAFL(v)	Returns the specific heat ratio of a fluid at a given temperature.

Function	Description
MF(“point name”)	Returns the local mass flow at the input point.
MIX(i1,i2,i3,.....,i10)	Returns the temperature from mixing two or more streams.
MMIX(i1,i2,i3,.....,i10)	Returns the mass flow rate from mixing two or more streams.
RE()	Returns Reynold’s number for a flow over a rotating body.
REL2ABS	Returns the absolute temperature of a fluid at a given relative temperature, radius, rotational speed, and fluid swirl velocity.
REL2ABS_SR	Returns the absolute temperature of a fluid at a given relative temperature, radius, rotational speed, and fluid swirl ratio.
REL2REL	Returns the relative temperature of a fluid at a given relative temperature, radius, rotational speed, and fluid swirl velocity for another rotational speed and fluid swirl velocity.
REL2REL_SR	Returns the relative temperature of a fluid at a given relative temperature, radius, rotational speed, and fluid swirl ratio for another rotational speed and fluid swirl ratio.
SA(i)	Returns the convecting area of a stream.
SMO(i)	Returns the outlet mass flow of the stream.
SP(i)	Returns the maximum pressure of the stream.
SPECFL(v)	Returns the constant pressure specific heat of a fluid at a given temperature.
SSV(i)	Returns the swirl velocity of the stream.
STI(i)	Returns the stream inlet temperature.
STMO(i)	Returns the outlet metal temperature of the stream.
STO(i)	Returns the stream outlet temperature.
TEMPF(“point name”)	Returns the local metal temperature of the fluid at the input point.
VA(i)	Returns the convecting area of a void.
VISCFL(v)	Returns the viscosity of a fluid at a given temperature.
VP(i)	Returns the maximum pressure of the void.
VSV(i)	Returns the maximum swirl velocity of the void.
VT(i)	Returns the maximum temperature of the void.
ZA(i)	Returns the convecting zone area.
ZP(i)	Returns the maximum pressure of the convecting zone.
ZSV(i)	Returns the maximum swirl velocity of the convecting zone.
ZT(i)	Returns the maximum temperature of the convecting zone.

User-defined functions for expressions

What is it?

NX can now dynamically evaluate user-defined functions for expressions. Dynamically evaluated functions update throughout an analysis.

User-defined functions are also known as plug-in functions. In earlier versions, NX could not dynamically evaluate plug-in functions. Because of this, their output was fixed throughout an analysis.

Materials

Multiple Fluid material

What is it?

The new **Multiple Fluid** material type lets you define a primary fluid material and up to five additional fluid materials.

The thermal solver uses the specified primary fluid material properties during computation. The additional fluid material number can be passed as an argument to some CAE functions, such as fluid plug-ins, and also to user-defined functions. See User-defined functions for expressions for more information on plug-in functions.

You define the **Multiple Fluid** material type in the following loads and modeling object:

- **Thermal Convecting Zone** load
- **Thermal Stream** load
- **Void Non-Geometric Element** modeling object

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics or NX Thermal and Flow as the specified solver.

Defining the multiple fluid material

Command Finder	**Manage Materials**
Location in dialog box	**Type** list→**Multiple Fluid**→ **Create**

Specifying the multiple fluid material in loads

Command Finder	**Thermal Convecting Zone** or **Thermal Stream**
Simulation Navigator	Right-click the **Load** container node→**New Load**→**Thermal Convecting Zone** or **Thermal Stream**
Location in dialog box	**Fluid Materials** list

Specifying the multiple fluid material in the modeling object

Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Void Non-Geometric Element**→**Create**→**Environment** group→**Fluid Materials** list

Norton-Bailey Power Law creep support in NX Multiphysics

What is it?

You can now use the Norton-Bailey Power Law creep model in Multi-step Nonlinear solutions when NX Multiphysics is the specified solver.

You select this model in the **Materials** dialog box, on the **Creep** page, using the **Temperature-Dependent Time Hardening (Norton-Bailey) Power Law (Type 301)** option. This option corresponds to the new `MATCRP` bulk data entry, which is an alternate method for selecting the Norton-Bailey creep model. It lets you define the coefficients of the power law equation as either constant or temperature dependent.

$$\varepsilon_e^c = A\sigma_e^B t^D$$

In the **Materials** dialog box, you define the A, B, and D coefficients of this creep model in the **Constant Multiplier (a)**, **SIGMA Exponent (b)**, and **Hardening Exponent (d)** boxes.

- To define creep at a single temperature, enter real values for the coefficients.
- To define creep as temperature dependent, define a table field for the coefficients. The table corresponds to the TABLEM1 bulk data entry.

Checking system units

A **Check Unit System** option is added to the new Type 301 power law as well as to the existing Time Hardening (Norton-Bailey) Power Law Type 300. This option checks for inconsistent units in the solution and the creep coefficients. NX checks when the solver input file is exported and alerts you if the solution units are not consistent with the creep coefficient units. NX cannot convert the creep coefficient units when it exports the solver input file.

Controlling creep effects

For NX Multiphysics, you can now turn off creep effects at the solution level and subcase level. For more information, see Controlling plasticity and creep effects.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver and an active Multi-step Nonlinear solution or NX Nastran as the specified solver and an active SOL 601,106 Advanced Nonlinear Statics solution
Command Finder	**Manage Materials**
Location in dialog box	**Create Material** →**Creep** page

Samcef damage properties

What is it?

The Samcef damage material models predict the fibers breaking, the matrix cracking, and the de-cohesion between fibers and the matrix. Delamination between two plies is also taken into account.

When you create orthotropic materials, you can use options on the new **Damage** page in the **Orthotropic Material** dialog box to model **Intra-laminar failure**. You can define a progressive damage model for a general orthotropic ply, for an enhanced ply (NCF), as well as for woven fabrics.

For these material models, you can specify properties such as:

- Nonlinear behavior in fiber direction.
- Damage evolution in the matrix, including coupling effects in shear and transverse directions.
- Plasticity behavior in the matrix.

You can also use the **Manage Materials** dialog box to create **Damage Interface** materials for cohesive elements.

Where do I find it?

Damage Interface material

Application	Advanced Simulation
Prerequisite	Samcef as the specified solver.
Command Finder	**Manage Materials**
Location in dialog box	**Type** list→**Damage Interface**→**Create Material**

Damage material model for orthotropic materials

Application	Advanced Simulation
Prerequisite	Samcef as the specified solver.
Command Finder	**Manage Materials**
Location in dialog box	**Type** list→**Orthotropic**→**Create Material** →**Damage** page

Adding material properties to existing library definitions

What is it?

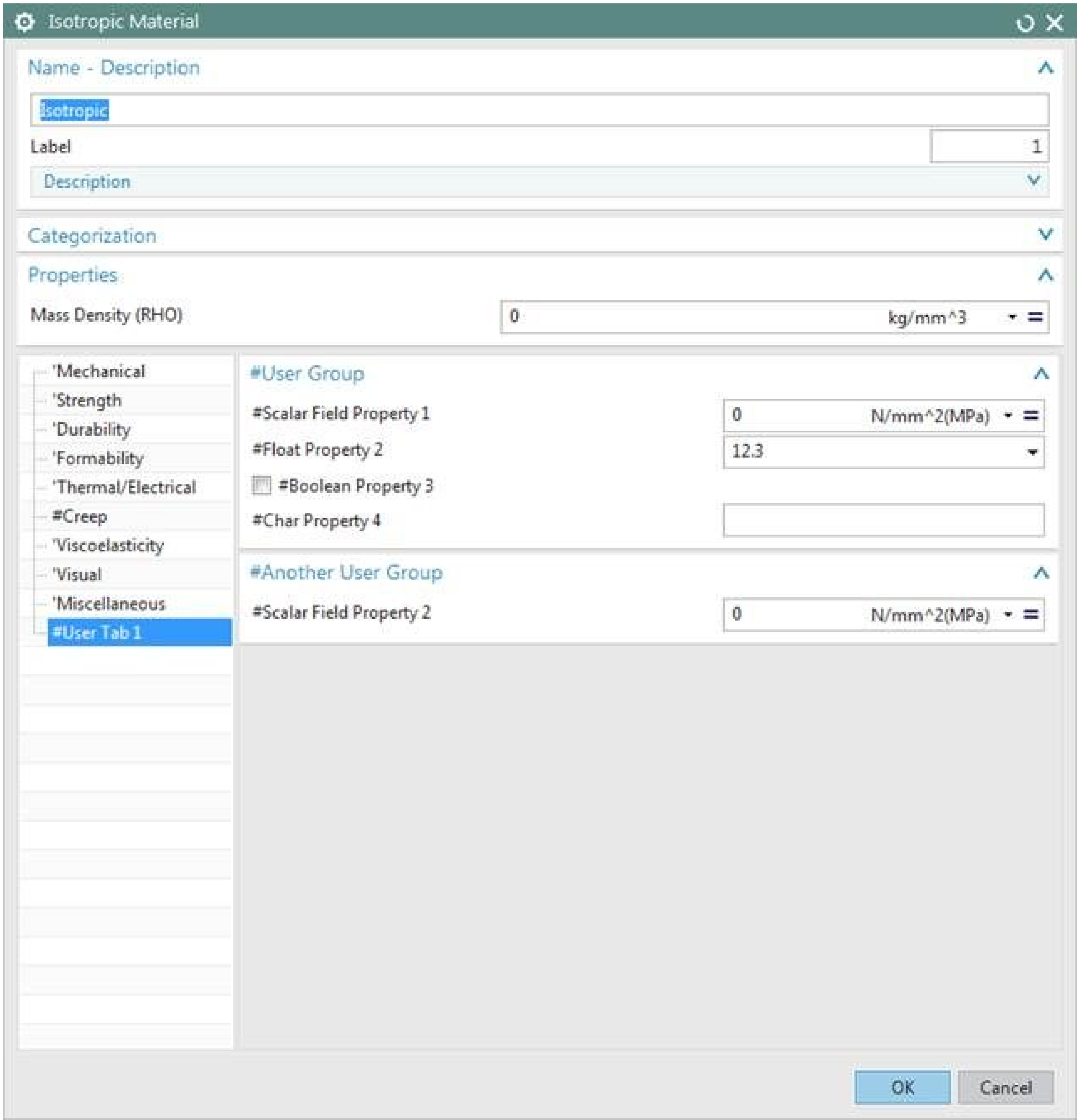
You can now add material properties to existing material library definitions using a correctly structured XML file. You can define these custom properties so that they appear in the materials dialog boxes on existing material property tabs or on new tabs. The custom properties always appear in new groups on the dialog box. If you define a property on a new tab and do not specify a name for the tab, the new tab is displayed as **DefaultTabName**.

You add material properties using an XML file with a .udmpx extension.

The DTD for this customization capability resides in the NX installation structure as *\..\ugii\materials\usermaterialsdefinitions.dtd*. The DTD defines the structure and legal elements of the XML file that you must provide.

Your users will be able to populate and edit these custom properties using the NX material dialog boxes, provided the NX session has the customization. To make the custom properties available to your users, you can place this XML file in any *UGOPEN_STARTUP_DIR* folder, such as *UGII_USER_DIR\\startup*, *UGII_SITE_DIR\\ startup*, and *UGII_VENDER_DIR\\startup*.

If a material defined and saved in NX contains a user-defined property, the property remains in the material object even when another NX session that does not have the customization opens the part.



User-defined properties added to a new tab

Support for plastic strain material nonlinearity in NX Multiphysics

What is it?

In the NX Multiphysics environment, you can now define a stress-dependent material with plastic strain properties. In the **Materials** dialog boxes, on the **Mechanical** page, a new **Type of Nonlinearity** option **PLSTRN** has been added.

This option corresponds to the `MATS1` bulk data entry with `TYPE=PLSTRN`. For more information, see *MATS1* in the *NX Nastran Quick Reference Guide*.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver An active Multistep Nonlinear solution
Command Finder	**Manage Materials**
Location in dialog box	**Create Material** →**Mechanical** page→**Stress-Strain Related Properties** group→**Type of Nonlinearity**=**PLSTRN**

Support for stress-strain related properties in orthotropic and anisotropic materials

What is it?

You can now specify stress-strain properties for orthotropic and anisotropic materials. The **Materials** dialog boxes include the **Stress-Strain Related Properties** group of options when you create a new orthotropic or anisotropic material.

For the corresponding syntax for each solver, see Stress-strain input as engineering or true stress-strain.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran, MSC Nastran, ANSYS, Abaqus, or LS-DYNA as the specified solver.
Command Finder	**Manage Materials**
Location in dialog box	**Create Material** →**Mechanical** page→**Stress-Strain Related Properties** group

Stress-strain input as engineering or true stress-strain

What is it?

You can now specify stress-strain data for nonlinear materials as either engineering stress-strain or true stress-strain.

When writing out the solver input file, NX converts the stress-strain data depending on the specified solver. When NX Nastran is the solver, NX additionally takes into account the type of solution and the setting of the **Type of Nonlinearity** option. The output stress-strain data is in the same units as the input stress-strain data. The units of Young’s Modulus should be consistent with the input stress-strain data.

In the **Materials** dialog boxes, on the **Mechanical** page, the **Stress-Strain Related Properties** group includes a new option, **Stress-Strain Input Data Type**, with these available types:

- **Engineering Stress-Strain** (default)
- **Engineering-Plastic Strain**
- **True Stress-Log Strain**
- **True Stress-Plastic Strain**
- **Undefined**

When they are migrated from earlier versions of NX, materials that contain stress-strain property data are set to the **Undefined** type automatically.

Conversion details

Solver	Solution type	Type of Nonlinearity option (corresponds to the TYPE field on `MATS1`)	NX converts to	Corresponding solver syntax
Nastran	SOL 601,106	NLELAST PLASTIC	There is no conversion. Stress-strain data is written out directly as specified in the material data.	`MATS1`, TYPE=NLELAST `MATS1`, TYPE=PLASTIC
		PLSTRN	Not supported. A fatal error is reported.	N/A
	SOL 106	NLELAST PLASTIC	Engineering Stress-Strain	`MATS1`, TYPE=NLELAST `MATS1`, TYPE=PLASTIC
		PLSTRN	Engineering Stress-Strain TYPE is changed to PLASTIC.	`MATS1`, TYPE=PLSTRN

Solver	Solution type	Type of Nonlinearity option (corresponds to the TYPE field on MATS1)	NX converts to	Corresponding solver syntax
NX Multiphysics	Multi-Step Nonlinear	NLELAST	Not supported. A fatal error is reported.	N/A
		PLASTIC	Engineering Stress-Strain	MATS1, TYPE=PLASTIC
		PLSTRN	Engineering Stress-Plastic Strain	MATS1, TYPE=PLASTIC
Abaqus	All	N/A	True Stress-Plastic Strain	*PLASTIC keyword
ANSYS	All	N/A	True Stress-Log Strain	Depends on the **Material plasticity model (stress-strain option)** that is selected in the **Solution** dialog box. • TB, MELAS (default) • TB, MISO • TB, KINH
LS-DYNA	All	N/A	True Stress-Plastic Strain	*MAT_PIECEWISE_LINEAR_PLASTICI keyword (constant temperature only)
Samcef	Nonlinear	PLASTIC	Samcef **Biot** , **Cauchy**, or **Kirchoff** laws	.MAT BIOT/CAUCHY/KIRCHOFF

Note

When **Stress-Strain Input Data Type** is set to **Undefined**, NX writes out the stress-strain data directly as specified in the material data and does not perform a conversion.

For example, if you enter engineering strain into NX for an ANSYS solution, NX performs the conversion from engineering strain to true strain as follows.

Type	Conversion
Engineering Strain → Log (True) Strain	$\varepsilon^l = \ln(1 + \varepsilon^T)$

Type	Conversion
Engineering Stress → True Stress	$\sigma = S\left(1 - \nu\varepsilon^{e} - \frac{1}{2}\varepsilon^{p}\right)^{-2}$ $\varepsilon^{e} = S/E$ $\varepsilon^{p} = \varepsilon^{T} - \varepsilon^{e}$

Where:

ε^{T} = Engineering Total Strain

ε^{e} = Engineering Elastic Strain

ε^{p} = Engineering Plastic Strain

S = Engineering Stress

v = Poisson’s Ratio

σ = True Stress

ε^{l} = Log (True) Strain

E = Young’s Modulus

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran, MSC Nastran, NX Multiphysics, Samcef, Abaqus, ANSYS, or LS-DYNA as the specified solver
Command Finder	**Manage Materials**
Location in dialog box	**Create Material** →**Mechanical** page→**Stress-Strain Related Properties** group

Geometry idealization and abstraction

Midsurface thickness source enhancements

What is it?

In the **Mesh Associated Data** dialog box, when you select **Midsurface** as the source of the thickness for a 2D mesh, you can use options on the new **Thickness Evaluation Type** list to control the location and manner in which the software evaluates the thickness:

- Select **At Every Node** to have the software perform ray casting operations to compute the thickness at each node in the mesh associated with the midsurface. For example, at a triangular element, the software casts rays from each of the element’s nodes to determine the thickness values.

- Select **At Element Centroid** to have the software perform a single ray casting operation at the centroid of each element to determine the thickness value. With this option, the software then assigns the same thickness value to all of the element's nodes. The **At Element Centroid** option offers slightly improved performance over the **At Every Node** option as the software performs fewer calculations.

- Select **Average Feature Thickness** to have the software extract the average thickness value for midsurface face pairs defined with the **Midsurface by Face Pairs**, **Offset Midsurface**, or the **User Defined Midsurface** commands. This option is the fastest way to evaluate the thickness as the software does not have to perform any ray casting calculations. It is the recommended option for constant thickness parts.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and work part
Command Finder	**Mesh Associated Data**
Location in dialog box	**Element Properties** group

Excluding geometry from abstraction

What is it?

Use the new **Freeze Geometry** command to designate faces that should be excluded from most automatic or manual geometry abstraction processes. When you use the **Freeze Geometry** command on a set of faces:

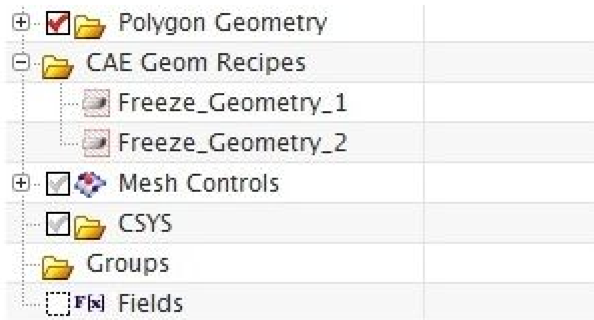
- The software does not perform any automatic abstraction or geometry cleanup operations on those faces during 2D or 3D meshing.

- You cannot select those faces for use with most polygon geometry commands. For example, you cannot use the **Stitch Edge** command to modify a face on which you have previously used the **Freeze Geometry** command.

You can continue to use the **Split Edge** and **Merge Edge** commands on faces that you select with the **Freeze Geometry** command.

New Freeze_Geometry node in the Simulation Navigator

NX stores the faces that you select as a **Freeze_Geometry** recipe in the **CAE Geometry Recipes** folder in the Simulation Navigator.



You can use the **Freeze_Geometry** nodes in the **Simulation Navigator** to:

- Rename the frozen geometry recipes.
- Display the locations of the frozen geometry.
- Delete selected frozen geometry recipes.
- Edit the faces in the frozen geometry recipes. You can add or remove faces from the recipe.

If you decide you want to later re-include frozen geometry in abstraction operations, delete the appropriate **Freeze_Geometry** node. Deleting the node will not change the polygon geometry or trigger an update of the FEM file; the faces are available for selection with manual abstraction commands and are included in any automatic abstraction operations.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the work part and displayed part
Command Finder	**Freeze Geometry**
Menu	**Insert→Model Cleanup→Freeze Geometry**

Surface Wrap commands support

What is it?

All commands on the **Surface Wrap** tab are now accessible from all solver environments supported in NX. These are:

- NX Thermal/Flow
- NX Electronic Systems Cooling
- NX Space Systems Thermal
- NX Multiphysics
- NX Nastran
- NX Nastran Design

- MSC Nastran
- Samcef
- Abaqus
- ANSYS
- LS-DYNA

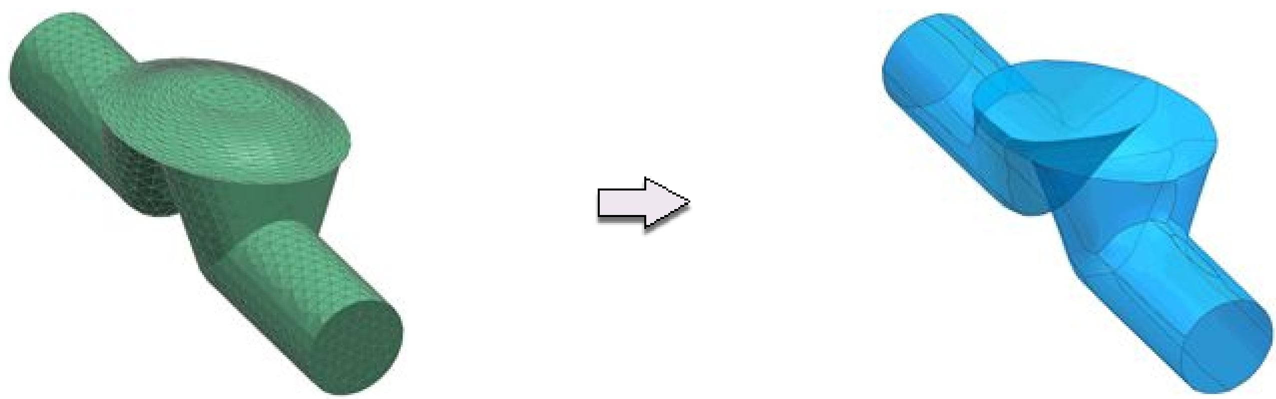
Why should I use it?

You can generate surface wrap bodies that you can use to create your simulation model in all solver environments.

Surface wrapping enhancement

What is it?

When you use the **Create Surface Wrap Body** command, you can now generate a surface wrap body by selecting imported meshes that are not associated to a polygon body. You can select 2D shell elements or 3D element free faces.



You can select both polygon bodies and elements in the same surface wrap recipe.

> **Note**
>
> Because imported meshes have no polygon edges, the **Snap to Source Edges** check box does nothing.

In previous releases, you could select only polygon bodies and the command name was called **Create Surface Wrap Fluid Domain**. NX now creates surface wrap bodies instead of fluid bodies.

Why should I use it?

You can generate surface wrap bodies from models for which you only have meshes.

Where do I find it?

Application	Advanced Simulation

Prerequisite	Import a meshed model into a FEM.
Command Finder	**Create Surface Wrap Body** 
Simulation Navigator	Right-click the **Surface Wrap Recipes** node→**New Surface Wrap Fluid Domain**
Location in dialog box	**Surface Wrap Boundaries** group

Synchronizing polygon body names to the CAD part

What is it?

Use the new **Synchronize to CAD Name** command in the **Simulation Navigator** to synchronize the names of bodies in a part file and the names of the corresponding polygon bodies. This allows you to rename selected polygon bodies to be consistent with the names of the corresponding bodies in the associated CAD part.

> **Note**
>
> You can only use the **Synchronize to CAD Name** command if the associated CAD part is loaded.

You cannot use the **Synchronize to CAD Name** command to rename:

- A polygon fluid body.
- A polygon body that is created from 2D elements or from 3D element free faces using the **Face from Mesh** command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and work part The CAD part associated with the FEM file loaded
Simulation Navigator	In the **Polygon Geometry** folder, right-click one or more **Polygon Body** nodes→**Synchronize to CAD Name**

Meshing

Adaptive meshing enhancements

What is it?

This release introduces a new adaptive meshing capability into NX. This capability builds upon the adaptive analysis solution process that was available in previous releases. These enhancements include:

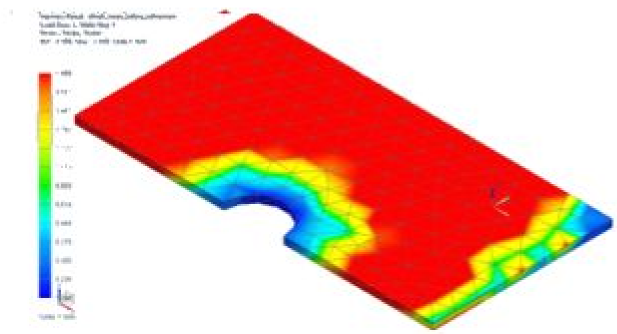
- An expanded range of supported analysis types, including both thermal and multi-physics analyses.

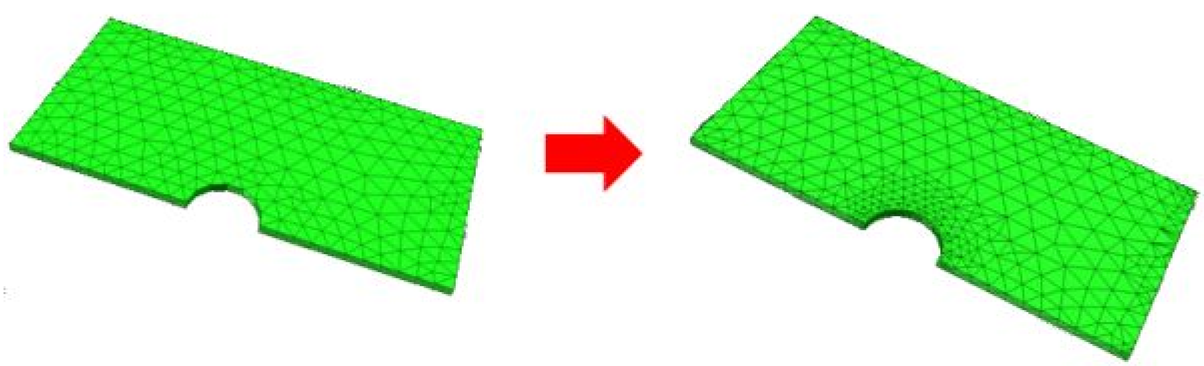
- Global and local refinement options for the mesh.
- The ability to exclude certain regions of the mesh from the adaptivity process.

In **Advanced Simulation**, you can use the **Adaptivity** solution process to set up and run adaptive meshing.

Benefits of the adaptive meshing process

Adaptive analysis automates the lengthy and repetitive process of running multiple finite element analyses with different mesh densities. The mesh refinement is determined by stress, strain energy, and temperature error estimates associated with element stress and strain discontinuities. Based on these error estimates, the software identifies critical regions on the model which become the target for further refinement of the mesh in subsequent iterations.





Supported solvers and analysis types

Solution type	Solvers	Notes
Linear statics analysis	NX Nastran, MSC Nastran, Abaqus, ANSYS	The mesh error is calculated by NX during the adaptivity process.
Thermal analysis (steady-state and transient)	NX Thermal	The mesh error is calculated by the NX Thermal solver during the adaptivity process.

<table>
<tr><th>Solution type</th><th>Solvers</th><th>Notes</th></tr>
<tr><td>Coupled thermal and structural (linear and non-linear) analyses</td><td>NX Multiphysics</td><td>The thermal and structural mesh errors are calculated independently.
• NX Thermal calculates the thermal error.
• NX Nastran calculates the structural error.
The thermal and structural solvers output independent mesh error values.</td></tr>
<tr><td>Mapping solution</td><td>NX Multiphysics</td><td>• You can include one mapping solution in the adaptive process.
• NX solves the mapping solution prior to solving the main solution in each iteration. The software then provides the results from the mapping solution to the primary solution.</td></tr>
</table>

Supported element types

You can use adaptive meshing to refine meshes of the following general types of elements:

- Triangular elements
- Quadrilateral elements
- Tetrahedral elements

You can include other types of elements in a model in which you use an **Adaptivity** solution process. However, the software ignores those element types during the refinement process. Other elements must be geometry-based if they are connected directly to adaptive meshes. You can also create contact or glue definitions to connect other types of elements to adaptive meshes.

Control over adaptive mesh parameters

You can use the options in the **Adaptivity Meta Solution** dialog box to control how NX remeshes the model during the adaptivity analysis. For example, the **Adaptivity** solution process now supports both local and global mesh refinement. You can use the **Meshing Method** list to control the scope of the error calculation and error analysis. For example:

- Select **Local** to have the software remesh only the regions where the specified error norms are high. With **Local** refinement, NX refines only the mesh on the faces where the error norms are high and the neighboring faces.
- Select **Global** to have the software remesh the entire part. With **Global** refinement, NX refines the mesh on all faces.

You can also control the number of iterations that NX performs during the analysis, as well as the maximum level of element refinement.

Excluding regions from adaptive refinement

You can use options in the new **Adaptivity Exclusion Zone** dialog box to define regions of the mesh that you want to exclude from adaptive refinement. The software treats these zones as converged regions that require no refinement. **Exclusion Zone** items are stored in the **Simulation Navigator** under the **Adaptivity** solution process node. For example, you may want to define an **Exclusion Zone** in a region to exclude regions of a part that are not critical to your current analysis.

Note

These regions may still remesh due to changes in the meshes on neighboring regions. However, changes to the mesh in an exclusion zone are not influenced by the error estimates in that region.

Accuracy parameters

In the **Adaptivity Setup** dialog box, you must define the accuracy parameters for the adaptivity process. These parameters, which are also known as error estimates, are the criteria that the software uses to determine whether the solution has converged. The accuracy parameters that the software uses depend upon the analysis type of your solution. For example:

- For structural analyses, NX Nastran generates strain energy norm errors, stress energy norm errors, or both error values.

- For thermal analyses, NX Thermal generates temperature error norm results.

For example, for NX Thermal analyses, the solver supports an output estimate of the local error in the temperature on nodes and elements at each output time. The software uses the maximum error at each node over all integration time steps, along with the time at which the maximum occurs, as the input for refining the mesh.

Adaptive meshing output

When the software completes the final iteration, it saves several different types of output. This output includes:

- The solution results, including the error listing for the final iteration only.

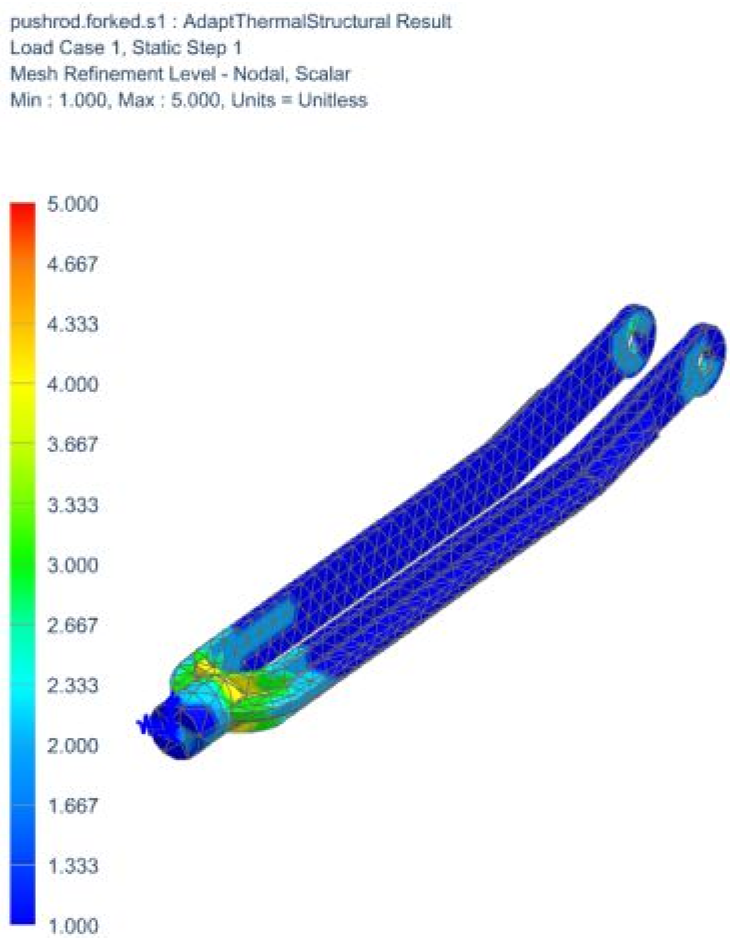
- The final mesh.

- A summary of the adaptive analysis that provides detailed information, including:
  - o The maximum actual value for each accuracy parameter at each iteration.
  - o The total number of iterations.
  - o The total number of elements and nodes in the mesh at each iteration.
  - o Whether or not the solution converged.

Viewing adaptivity results in NX Post Processing

After the **Adaptivity** iterations complete, you can review the results of the adaptive solution in the **Post-Processing Navigator**. For example, you can create contour plots of the strain energy error as well as the stress or temperature errors. You can also create contour plots of the mesh refinement level results.

When you create a contour plot of the mesh refinement level results, the software generates a banded display that indicates the level of refinement necessary at each node based on the computed error results from the adaptive analysis.



Where do I find it?

Adaptivity solution process

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part NX Nastran, MSC Nastran, NX Multiphysics, NX Thermal, Abaqus, or ANSYS as the specified solver
Simulation Navigator	Right-click the Simulation file→**New Solution Process**→**Adaptivity**
Menu	**Insert**→**Adaptivity Setup**

Solve an adaptivity solution

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part NX Nastran, MSC Nastran, NX Multiphysics, NX Thermal, Abaqus, or ANSYS as the specified solver An existing adaptivity solution
Simulation Navigator	Right-click the adaptivity solution→**Solve**

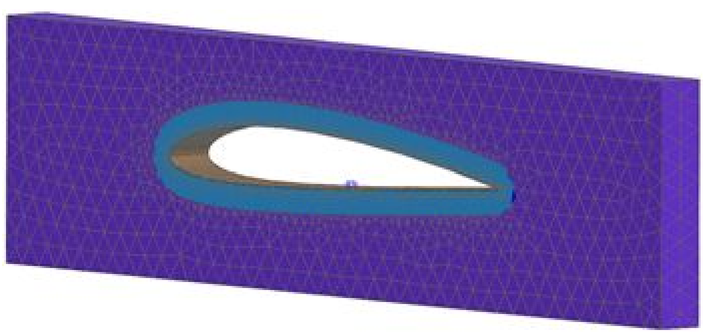
Adaptivity Exclusion Zone

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part NX Nastran, MSC Nastran, NX Multiphysics, NX Thermal, Abaqus, or ANSYS as the specified solver
Simulation Navigator	In the appropriate **Adaptivity** solution process node, right-click the **Exclusion Zones** folder→**New**

Boundary layer meshing in the FEM

What is it?

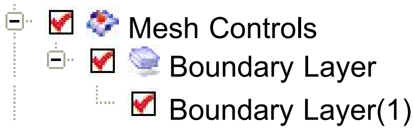
You can now create boundary layer meshes directly in the FEM.



Workflow

Step 1: Define a boundary layer mesh control on the polygon face from which you want to start the boundary layers. You need to specify:

- The number of element layers in the boundary layer mesh.
- The thickness of the first layer of elements in the boundary layer mesh.
- The method used to define the height of the boundary layer. You can either specify the total thickness of the boundary layer or the growth rate of the element thickness in the boundary layer. For example, a growth rate of 1.1 results in every successive layer being 10% thicker than the previous one.



Step 2: (Optional) Create a 2D mesh on the polygon face that has the boundary layer mesh control.

If you create a triangular surface mesh, the boundary layer mesh will contain wedge elements. If you create a quadrilateral surface mesh, the boundary layer mesh will contain hexahedral elements. If you do not create a 2D mesh, NX automatically generates a triangular mesh on the surface.

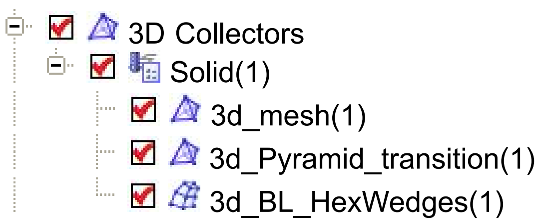
Note

You need to specify an element type that is linear.

Step 3: Create a 3D tetrahedral mesh on your polygon body.

Note

You need to specify an element type that is linear.



Results

NX creates up to five solid meshes:

- A boundary layer mesh composed of hexahedral elements and wedge elements. This mesh is always created. In the **Simulation Navigator**, this mesh is stored in the **3d_BL_HexWedges** nodes.
- A boundary layer mesh composed of tetrahedral elements. NX creates these elements as transition elements. In the **Simulation Navigator**, this mesh is stored in the **3d_BL_Tets** nodes.
- A boundary layer mesh composed of pyramid elements. NX creates these elements as transition elements. In the **Simulation Navigator**, this mesh is stored in the **3d_BL_Pyramids** nodes.
- A transitional mesh composed of pyramidal elements, to transition from hexahedral boundary layer elements or quadrilateral shell elements to the tetrahedral elements. In the **Simulation Navigator**, this mesh is stored in the **3d_Pyramid_transition** nodes.
- A 3D tetrahedral mesh that meshes the rest of the polygon body volume. This mesh is always created.

The number of solid meshes created depends on the complexity of your model and other mesh controls and meshes defined prior to creating the boundary layer mesh.

In previous releases, to create a boundary layer mesh, you needed to define a **Fluid Domain** simulation object in the Simulation file and solve the solution to display the boundary layer mesh in the **Post Processing Navigator**. You could not view the boundary layer mesh before you solved your solution.

Why should I use it?

Use the boundary layer meshing for CFD analysis and acoustics modeling.

Where do I find it?

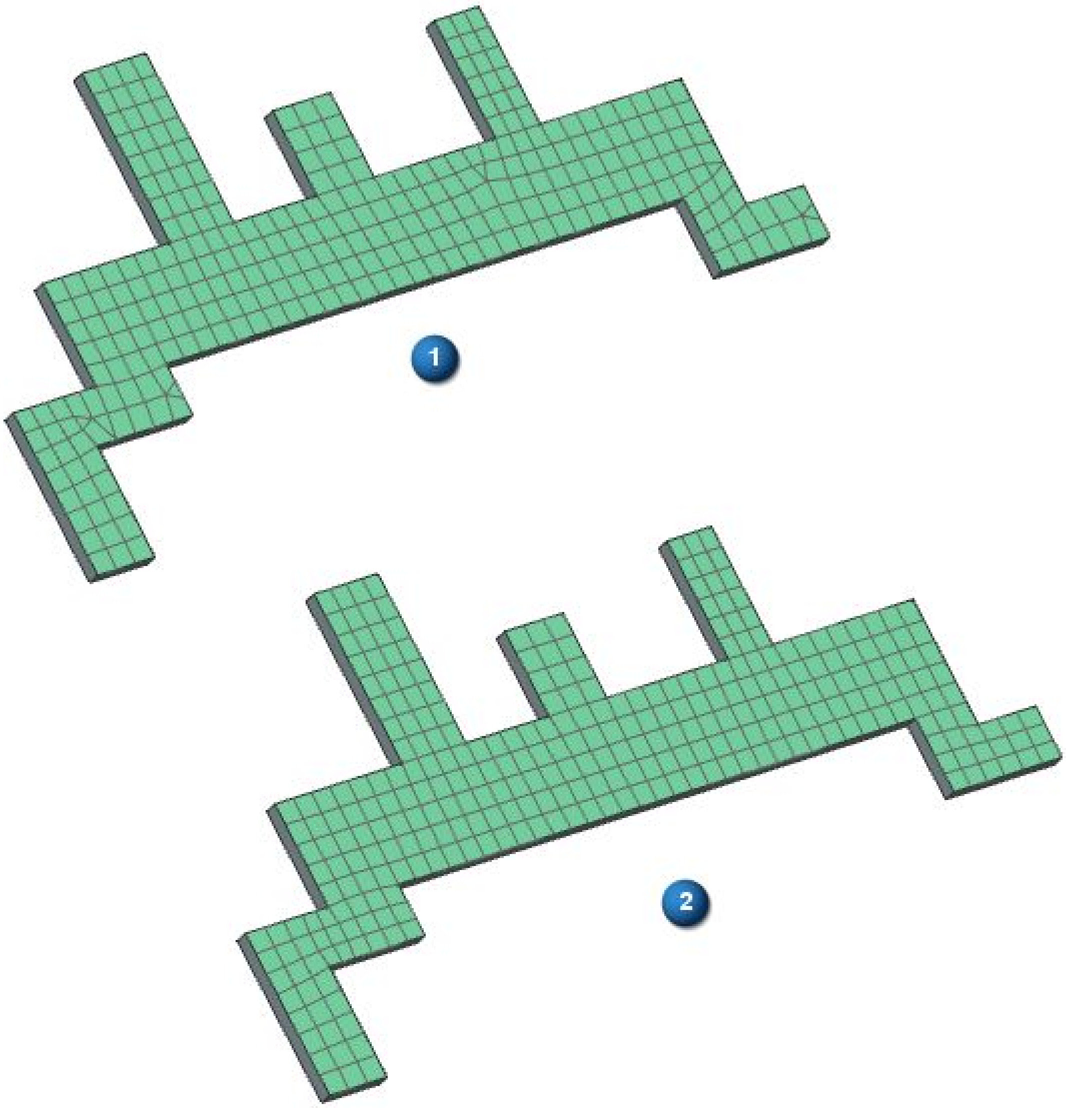
Application	Advanced Simulation
Prerequisite	A FEM as the work part
Command Finder	**Mesh Control**
Simulation Navigator	Right-click the **Mesh Controls** node→**New Mesh Control** or **New Mesh Control Definition**
Graphics window	Right-click a polygon face→**Mesh Control**
Location in dialog box	**Density Type**→**Boundary Layer**

Multi-block decomposition technique for 2D meshing

What is it?

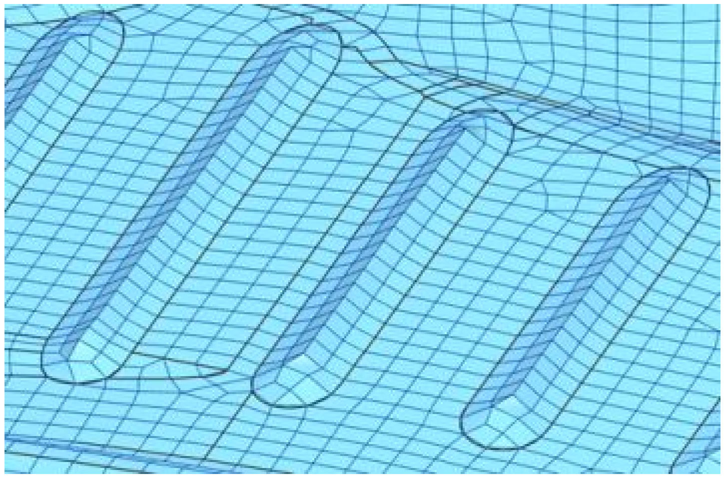
Use the new **Attempt Multi-Block Decomposition** option in the **2D Mesh** dialog box to improve quadrilateral meshes on largely orthogonal surfaces (surfaces with a number of 90° corners). When you select this option, NX breaks up faces into a cluster of convex regions and then generates meshes on each convex region. In general, the resulting meshes are more orthogonal in nature. If you select both the **Attempt Multi-Block Decomposition** and the **Attempt Free Mapped Meshing** options, NX creates a structured mesh on more regions of your model.

The graphics show the difference between a surface meshed with quadrilateral elements (1) and the same set of surfaces meshed using the new **Attempt Multi-Block Decomposition** option (2).

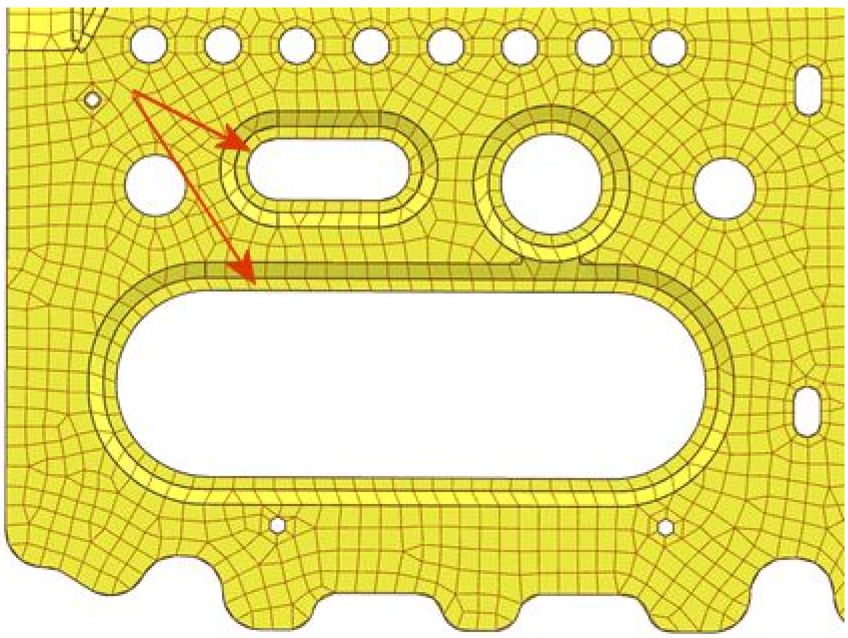


Additionally, when you select **Attempt Multi-Block Decomposition**, **Attempt Free Mapped Meshing**, and the new **CAD Curvature Abstraction** options, the software uses a methodology to analyze surfaces and identify certain types of features, such as narrow bead and annular surfaces. NX is then able to produce a better, more structured mesh in those regions.

With these improvements, the software is now able to, for example, capture details, such as the crests of beads, while creating a mapped mesh along the body of the bead, as shown in the following graphic.



NX can now also produce an all quadrilateral mesh along annular faces, as shown in the following graphic.



Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and work part
Command Finder	**2D Mesh**
Menu	**Insert→Mesh→2D Mesh**

Improved feature preservation and abstraction in 2D meshing

What is it?

Use the new **CAD Curvature Abstraction** option in the **2D Mesh** dialog box to improve quadrilateral meshes on relatively thin-walled parts with curvature, such as automotive body panels. When you select this option, the software uses a new methodology to abstract the geometry in your model. This methodology uses a curvature analysis to improve the quality of the quadrilateral elements in your model by preserving the representation of part features during the geometry abstraction process.

In previous releases, the geometry abstraction process treated an entire polygon body as a single entity during the abstraction process. In NX 10, the abstraction process analyzes groups of adjacent faces that have similar curvature. Where possible, it merges those individual faces together. For smaller, sliver faces within the model, such as tiny faces along fillets, the software more aggressively merges those faces with adjacent faces. This new process aims to preserve the boundaries of features while eliminating very small faces that are interior to features. This results in fewer small elements in the mesh that violate the specified **Minimum Element Length** value.

Note

The **CAD Curvature Abstraction** option is valid only for free meshes with quadrilateral elements.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and work part The associated CAE part loaded
Command Finder	**2D Mesh**
Menu	**Insert→Mesh→2D Mesh**
Location in dialog box	**Model Cleanup** group

Support for 2D seed mesh creation in NX Multiphysics

What is it?

In the NX Multiphysics environment, when you create a mesh for a structural analysis, you can now create 2D seed meshes to control the placement of nodes on subsequent 3D meshes. In the **2D Mesh** or the **2D Mapped Mesh** dialog box, you can select either the **Linear Quadrilateral Seed** or **Linear Triangle Seed** options from the **Type** list to create the seed mesh. These seed meshes have the following characteristics:

- They can only be used to create solid meshes.
- They do not have any associated physical properties or mesh associated data.
- They are not written out to the solver input file when you export or solve your model.

In previous releases, you could create only 3D meshes for models in which NX Multiphysics was the specified solver.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and work part NX Multiphysics as the specified solver **Structural** as the specified analysis type
Command Finder	**2D Mesh** or **2D Mapped Mesh**
Menu	**Insert→Mesh→2D Mesh**

Updated support for 3D manual swept meshes

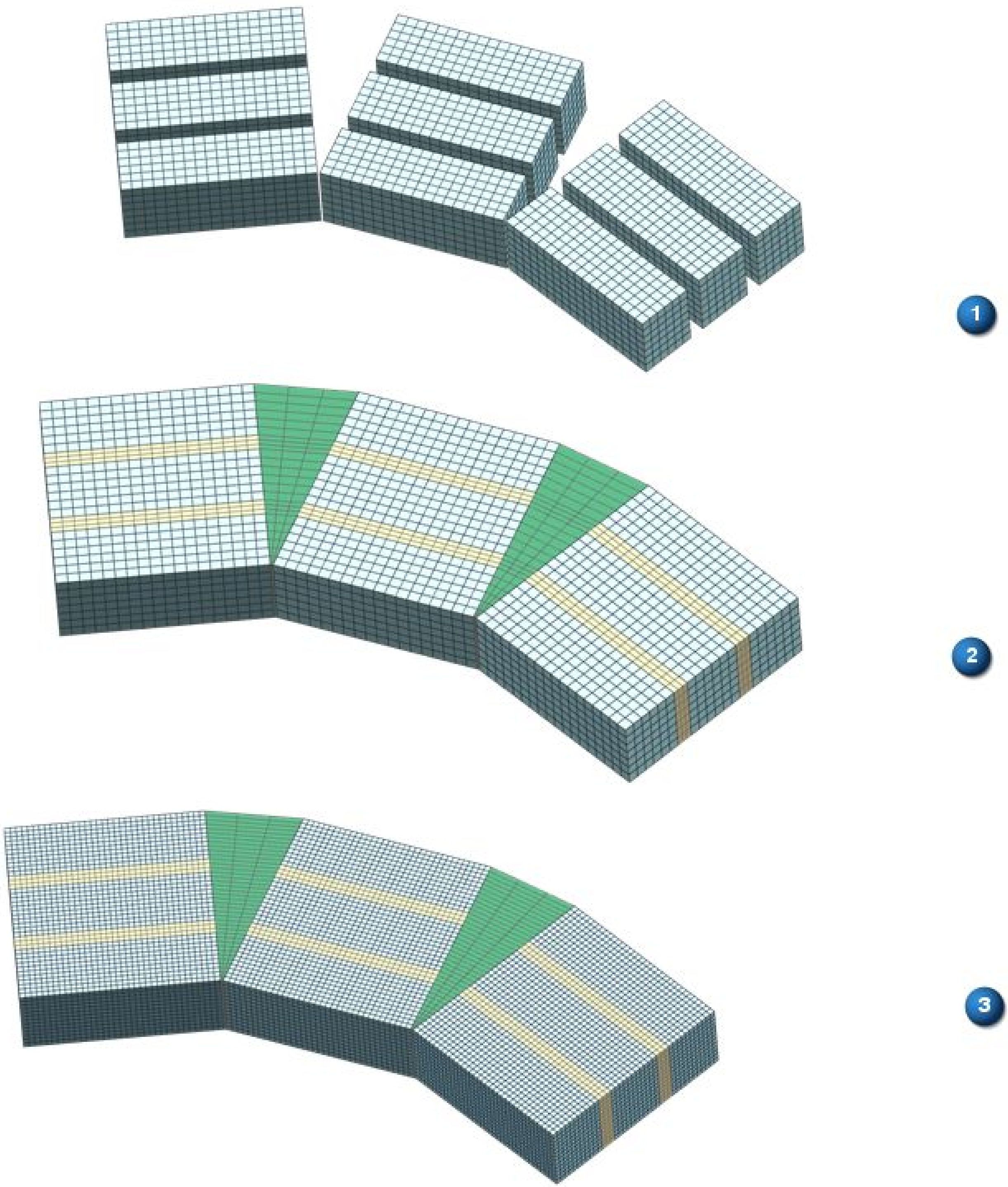
What is it?

You can now use the **3D Swept Mesh** command to automatically or manually create hexahedral meshes between selected faces. Use the new **Automatic Between** and **Manual Between** options in the **Type** list to sweep a structured 3D mesh between two solid meshes. For example, you can use these options to create a mesh of brick or wedge elements even if you do not have any geometry to define a body.

Swept meshes that you create with the **Automatic Between** and **Manual Between** options can update either as the geometry updates, or as the parameters of the neighboring meshes change. NX now creates a mesh recipe for manual hexahedral meshes in the **Simulation Navigator**. You can use this recipe to modify the swept mesh. NX also uses this mesh recipe to update the manual swept mesh when changes occur to adjacent meshes or in the underlying geometry.

In previous releases, you used the **3D Sweep Between** command to manually create hexahedral meshes. However, those meshes were entirely FE-based and did not update.

In the example, (1) shows an assembly of blocks that are all meshed separately. (2) shows swept meshes created using the **Manual Between** option. (3) shows how the swept meshes update when the size of the meshes on the blocks update.



Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part

Command Finder	**3D Swept Mesh**
Menu	**Insert→Mesh→3D Swept Mesh**

Contact mesh usability improvements

What is it?

The **Contact Mesh** dialog box has been redesigned to improve its usability and workflow. You use the **Contact Mesh** command to create point-to-point contact between two edges or a portion of two edges that you define. The type of elements that you can create between the selected edges depends on your specified solver. For example, in the NX Nastran environment, you can create CGAP elements, while in the Abaqus environment you can create GAPUNI elements.

Additionally, the **Contact Mesh** command now supports journaling.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part NX Nastran, MSC Nastran, Abaqus, or ANSYS as the specified solver.
Command Finder	**Contact Mesh**
Menu	**Insert→Mesh→Contact Mesh**

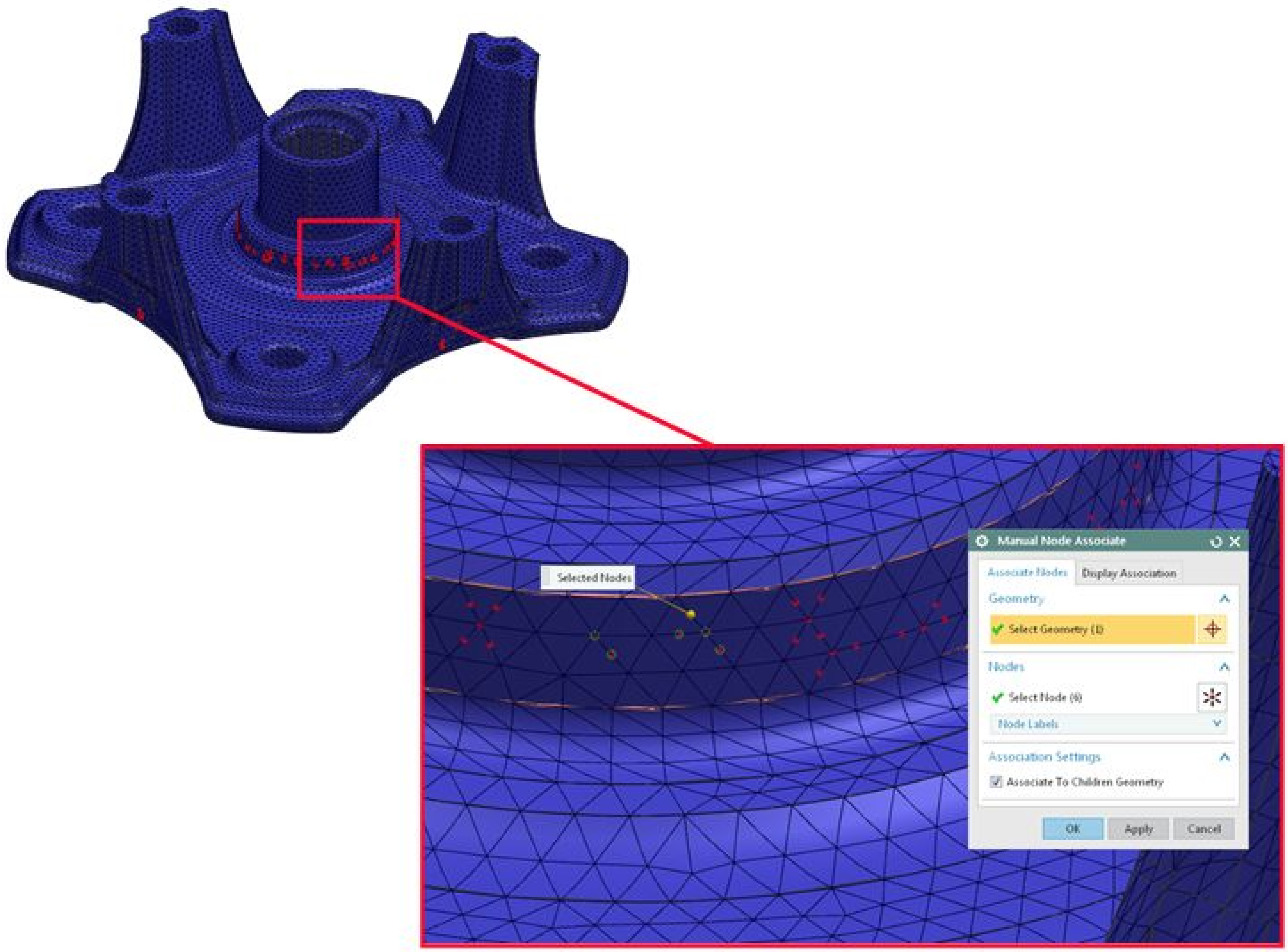
Manually associating nodes to geometry

What is it?

You can now use the **Manual Node Association** command to manually associate nodes in an orphaned mesh to selected geometry. This command gives you more granular control over which nodes are associated to specific geometry. For example, you can use **Manual Node Association** to associate a selected set of orphan nodes to a selected edge.

As a best practice, you should first use **Node Association** to have the software automatically associate as many orphan nodes as possible to nearby polygon geometry. You can then use the **Manual Node Association** command to associate any remaining orphan nodes.

> **Note**
>
> **Manual Node Association** controls only the geometry association between a node and geometry. It does not change the location of a node.



Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part
Command Finder	**Manual Node Association**
Menu	**Edit→Mesh→Manual Node Association**

Saving full morphing data

What is it?

When you create a FEM file, you can use the new **Save Full Morphing Data** option in the **New FEM** dialog box to control whether the software stores comprehensive data about modifications to the underlying CAD geometry. If you plan to use morphing and not the **FEM Update** command to update the meshes in your model in subsequent NX sessions, select this option because it stores additional data about changes to the CAD geometry. If you use the morphing commands in a subsequent NX session, this additional data allows NX to produce higher quality morphing results.

Note

The **Save Full Morphing Data** option does not have any impact on the quality of morphing results when you modify the CAE geometry and morph an associated mesh in the same NX session.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A part displayed
Simulation Navigator	Right-click the part file→**New FEM**
Location in dialog box	**Mesh Morphing** group

Geometry filtering for non-circular holes with Mapped Hole mesh controls

What is it?

When you use the **Filter** option in the **Mesh Control** dialog box to create a **Mapped Hole** mesh control, you can select the **Allow Non Circular Holes** check box to have the software search for both circular and non-circular holes that meet the criteria you specify. In previous releases, the automatic geometry selection filtering for **Mapped Hole** mesh controls searched only for circular holes.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and work part
Command Finder	**Mesh Control**
Menu	**Insert**→**Mesh**→**Mesh Control**

Export Mesh to Solver option more widely available

What is it?

You can now use the **Export Mesh to Solver** option to control whether manually created meshes are written out to the solver input file when you export or solve your model. This option is now available for the following manual element creation commands when you create 2D elements:

- **Element Create**
- **Element Extrude**
- **Element Revolve**
- **Element Copy and Translate**
- **Element Copy and Project**

- **Element Copy and Reflect**

In previous releases, meshes that you created using these commands were always written out to the solver input file.

Additionally, the **Export Mesh to Solver** option is now available in the **Mesh Associated Data** dialog box for 2D meshes.

Where do I find it?

Manual element creation commands

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part
Command Finder	**Element Create**, **Element Extrude**, **Element Revolve**, **Element Copy and Translate**, **Element Copy and Project**, **Element Copy and Reflect**
Menu	**Insert→Element**
Location in dialog box	**Destination Mesh** group

Mesh Associated Data

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part
Simulation Navigator	Right-click a 2D mesh→**Mesh Associated Data**

Improvements for editing multiple meshes

What is it?

You can now edit multiple meshes and have the software retain the existing element type of each mesh. This allows you to make selected changes to meshes that have the same topology without affecting the specified element type settings. Now, when you select multiple meshes in the **Simulation Navigator** and use the right-click **Edit Mesh** option, you can use the new **Retain Type** option in the **Edit Mesh** dialog box to indicate that you do not want to change the type of elements used in the selected meshes.

- The **Retain Type** option is available when you select multiple meshes with different element types or multiple meshes which use different mesh collectors.

- When you select the **Retain Type** option, the **Destination Collector** options in the **Edit Mesh** dialog box become insensitive. NX uses the existing collector for each selected mesh.

Where do I find it?

Application	Advanced Simulation

Prerequisite	A FEM file as the displayed part and the work part
Simulation Navigator	Right-click one or more meshes→**Edit**

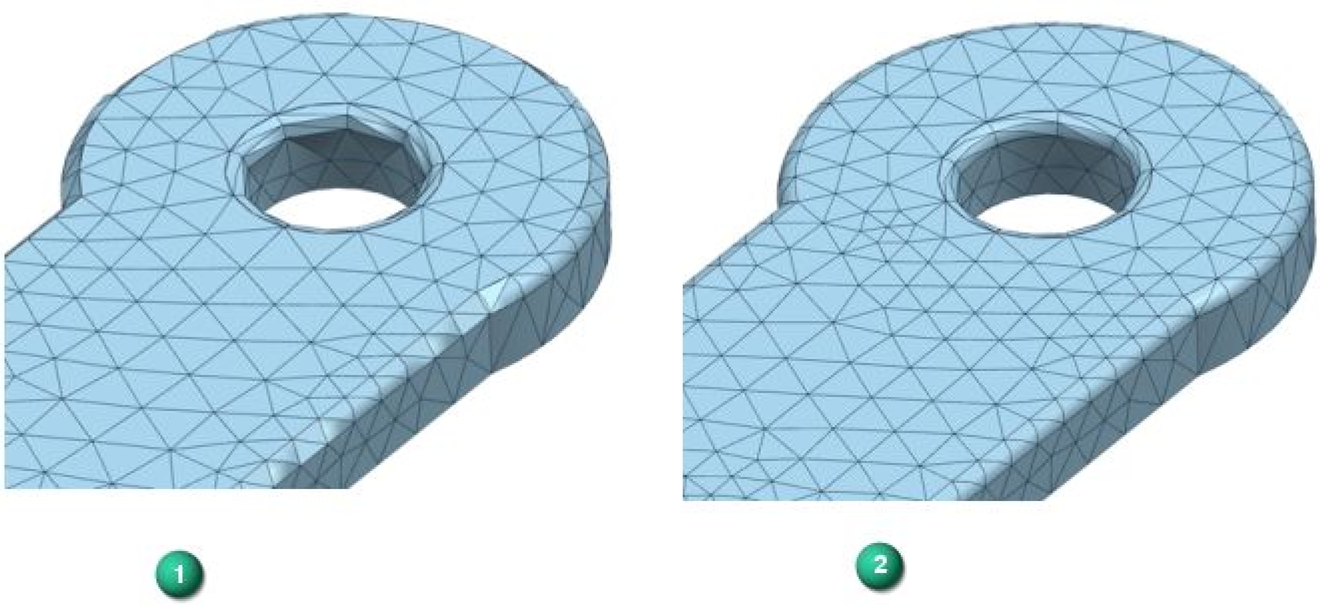
Midnode placement changes for tetrahedral elements

What is it?

This release includes improvements to the process that NX uses to place midnodes on the edges of parabolic tetrahedral elements. In previous releases, if the software detected that an element violated the specified **Max Jacobian** value specified in the **3D Mesh** dialog box at the element's Gauss points, NX straightened the element's edges.

In NX 10, some small enhancements have been made to the element edge straightening algorithm. The software now iteratively moves the midnodes off the geometry to try to preserve the curvature of the underlying geometry. Instead of immediately reverting to a straight element edge, the software incrementally straightens the edge by moving the midnode along a line between the original location and a final straight location. The software first moves the node 25% of the distance along that line, then 50%, then 75%, then 100%. If the element passes the Jacobian criteria with the node at the 25%, 50%, or 75% location, then no further straightening occurs. As a result of this process, the nodes remain closer to the underlying geometry while still satisfying the quality criteria.

The following graphic shows the subtle differences between a 3D mesh generated in NX 9 (1) and a 3D mesh generated in NX 10 (2).



Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part
Command Finder	**3D Tetrahedral**

Menu	**Insert→Mesh→3D**

Jacobian calculation changes for Nastran tetrahedral elements

What is it?

The Jacobian element quality check in the NX Nastran and MSC Nastran environments now calculates the Jacobian values for tetrahedral elements only at their Gauss points. In previous releases, NX calculated an element's Jacobian values at its Gauss points and at its corner nodes.

Neither NX Nastran nor MSC Nastran fails to solve due to Jacobian errors at an element's corner nodes; Nastran only fails to solve if Jacobian errors occur at an element's Gauss points. Consequently, in the Nastran environments, the **Element Quality Checks** command no longer evaluates Jacobian values at an element's corner nodes.

Note

NX Nastran and MSC Nastran still list warnings for high Jacobian values at an element's corner nodes to the .f06 file. To view these warnings, you can review the .f06 file after the solve.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part NX Nastran or MSC Nastran as the specified solver
Command Finder	**Element Quality Checks** 
Menu	**Analysis→Finite Element Model Check→Element Quality**

Creating plane strain and plane stress elements parallel to axisymmetric plane

What is it?

You can now create plane strain and plane stress elements on a plane that is parallel to the axisymmetric plane, we well as on the axisymmetric plane itself. To create these elements on a parallel plane, use a mesh creation command, such as **2D Mesh**, or manual element creation commands, such as **Element Create** and **Element Translate**. This change gives you greater flexibility when you create models. In previous releases, you could create plane strain and plane stress elements directly on the axisymmetric plane only.

Where do I find it?

Application	Advanced Simulation

Prerequisite	A FEM file as the displayed part and the work part NX Nastran, NX Multiphysics, Abaqus, ANSYS, NX Electronic Systems Cooling, NX Thermal and Flow, or NX Space Systems Thermal as the specified solver **Axisymmetric Structural** or **Axisymmetric Thermal** as the analysis type, or **Structural** or **Thermal** as the analysis type with the **2D Solid Option** set to any option other than **None**

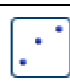
Changes to 0D element creation

What is it?

Beginning in this release, point-based meshes that you create on a face or an edge with the **0D Mesh** command now follow the underlying mesh on the edge or face. For example, if you use the **0D Mesh** command to define a series of elements along an edge, NX generates those 0D elements only after you generate a beam mesh along that edge. Similarly, if you use **0D Mesh** to define an element at a location on a face, NX generates the 0D element after you create the mesh on that face.

Additionally, NX no longer creates a mesh point when you create a 0D element. In previous releases, NX created a mesh point for every node that was associated with a 0D element. This caused performance issues during the mesh update process when changes occurred to the underlying CAD geometry.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part
Command Finder	**0D Mesh** 
Menu	**Insert→Mesh→0D Mesh**

Batch Meshing enhancements

What is it?

The NX Batch Meshing utility has been updated to include many of the enhancements to the 2D meshing capabilities from this release. The following table details the new batch meshing parameters and their use.

Parameter name	Parameter file entry	Description
CAD curvature abstraction	`CAD_curvature_abstraction`	Improves quadrilateral meshes on relatively thin-walled parts with curvature, such as automotive body panels. With this parameter, the software uses a new methodology to abstract the geometry in your model. When you turn this parameter on, the software more aggressively merges smaller, sliver faces with larger, adjacent faces. This results in fewer small elements in the mesh that violate the specified **Minimum Element Length** value. **Note** The **CAD Curvature Abstraction** option is valid only for free meshes with quadrilateral elements. For more information, see Improved feature preservation and abstraction in 2D meshing.
Minimum element length	`minimum_feature_length`	If you use the `CAD_curvature_abstraction` parameter, you must also use the `minimum_feature_length` parameter to specify a target minimum element length value. The software uses this value to determine which small faces it should merge during the abstraction process.
Multi Block decomposition	`multi_block_decomposition`	Improves quadrilateral meshes on largely orthogonal surfaces (surfaces with a number of 90° corners). When you turn this parameter on, NX breaks up faces into a cluster of convex regions. The software then generates meshes on each convex region. In general, the resulting meshes are more orthogonal in nature.

Parameter name	Parameter file entry	Description
		For more information, see Multi-block decomposition technique for 2D meshing.

Boundary Conditions

Load sets

What is it?

You can define and organize the loads in NX Nastran, MSC Nastran, and NX Multiphysics simulation files using load sets. The load sets are stored in the **Solver Sets** folder in the Simulation Navigator. You can also create the load set in the active solution or subcase.

You can create loads using a right mouse button command in the appropriate type of load set, or you can use the drag and drop method to add existing loads.

The following types of load sets are available:

- **Static Load Set**
- **Frequency Excitation Load Set**
- **Transient Excitation Load Set**
- **Acoustic Source Load Set**
- **Temperature Set**

Using Folders

You can add folders of loads to load sets. If the folder contains loads other than those accepted by the particular load set type, NX adds the folder, but disables the unaccepted loads.

Supported solution sequences

The following table shows the types of solutions that are available for each Nastran solution type.

Load set type	Solutions
Static Load Set	All Nastran and NX Multiphysics solution types
Frequency Excitation Load Set	SOL 108, 111, 200
Transient Excitation Load Set	SOL 109, 112, 129, 159, 200, 401, 601, 701
Acoustic Source Load Set	SOL 108, 111
Temperature Set	All Nastran and NX Multiphysics solution types

Static Load Set

What is it?

You can use a **Static Load Set** to specify the structural loads for use in an NX Nastran, MSC Nastran, or NX Multiphysics analysis.

There are two types of static load sets:

- **Applied load**
- **Enforced Motion**

The type of static load set you select limits the type of loads that you can add. For example:

- If you select **Enforced Motion**, you can only add enforced motion loads.
- You cannot add bolt pre-loads to a static load set.
- If you select **Applied Load**, you can add all other structural load types.

You can add gravity to an **Applied Load** static load set; however, you cannot combine gravity with other types of loads in the same load set. NX Nastran requires the load set identification number (SID) assigned to gravity loads to be unique and different from the SID assigned to all other loads.

Corresponding Nastran Syntax

A **Static Load Set** corresponds to the NX Nastran `LOAD` case control command.

The **Label** field in the **Static Load Set** dialog box assigns the `SID` for NX Nastran.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver
Simulation Navigator	Right-click **Solver Sets** → **New** → **Static Load Set**

Transient Excitation Load Set

What is it?

You use a **Transient Excitation Load Set** to define a time-dependent dynamic load or enforced motion load for use in a transient response analysis.

The transient excitation load set defines a load as a function of time, using either an expression or a field. You can also define a dynamic load time delay directly in the **Transient Excitation Load Set** dialog box.

Adding loads to a Transient Excitation Load Set

You cannot add individual loads to a dynamic load set, but you can add a static load set, and add loads to that.

- You can add an **Applied Load** static load set only to a transient excitation load set of type **Applied Load**.

- You can add an **Enforced Motion** type of static load set only to a transient excitation load set of type **Enforced Displacement**, **Enforced Velocity**, or **Enforced Acceleration**.

- The type of motion created by an **Enforced Motion Load** is specified by the type of transient excitation load set into which you place the static load.

Corresponding Nastran Syntax

The **Transient Excitation Load Set** corresponds to the NX Nastran `TLOAD1` case control command.

The **Label** field in the **Transient Excitation Load Set** dialog box assigns the `SID` or load set identification number for NX Nastran.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver
Simulation Navigator	Right-click **Solver Sets** → **New** → **Transient Excitation Load Set**

Frequency Excitation Load Set

What is it?

You use a **Frequency Excitation Load Set** to define a frequency-dependent dynamic load for use in a frequency response analysis.

The excitation you create is applied over all frequencies for all degrees of freedom. You can define the value of the excitation as a function of frequency, using any of the following:

- A complex expression

- A field

- Magnitude and phase

You can also define dynamic load phase leads and time delay in the **Frequency Excitation Load Set** dialog box.

Adding loads to a Frequency Excitation Load Set

You cannot add individual loads to a dynamic load set, but you can add a static load set, and add individual loads to that.

- You can add an **Applied load** type of static load set only to a frequency excitation load set of type **Applied load**.

- You can add an **Enforced Motion** type of static load set only to a frequency excitation load set of type **Enforced Displacement**, **Enforced Velocity**, or **Enforced Acceleration**.

- The type of motion created by an **Enforced Motion Load** is specified by the type of frequency excitation load set into which you place the static load.

Corresponding Nastran Syntax

The **Frequency Excitation Load Set** corresponds to the NX Nastran `RLOAD1` case control command.

The **Label** field in the **Frequency Excitation Load Set** dialog box assigns the `SID` or load set identification number for NX Nastran.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver
Simulation Navigator	Right-click **Solver Sets** → **New** → **Frequency Excitation Load Set**

Acoustic Source Load Set

What is it?

You use an **Acoustic Source Load Set** to define the acoustic source loads for use in an acoustic or vibro-acoustic analysis.

The acoustic source load set defines the fluid properties and the power of the acoustic excitation as a constant value or as a function of the frequency. You can also define dynamic load phase leads and time delay in the **Acoustic Source Load Set** dialog box.

The only type of load that you can add is a **Darea Acoustic Source** load.

Corresponding Nastran Syntax

The **Acoustic Source Load Set** corresponds to the `ACSRCE` bulk data entry.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran or MSC Nastran as the specified solver SOL 108 or SOL 111 as the specified solution type **Acoustic** or **Vibro-Acoustic** as the specified analysis type Mesh of Acoustic Fluid elements
Simulation Navigator	Right-click **Solver Sets** → **New** → **Acoustic Source Load Set**

Temperature Set

What is it?

You use a **Temperature Set** to specify the temperatures for use in an NX Nastran, MSC Nastran, or NX Multiphysics analysis.

Temperature sets have 3 types:

- **Initial/Stress Free Temperature** defines the initial temperature distribution.

  This type corresponds to the `TEMPERATURE(INITIAL)` case control command.

- **Material Temperature** is used to determine temperature-dependent material properties.

  This type corresponds to the `TEMPERATURE(MATERIAL)` case control command.

- **Temperature Load** updates the temperature-dependent material properties, and defines the temperatures used for thermal loading.

  This type corresponds to the `TEMPERATURE(LOAD)` case control command.

  For NX Multiphysics, this type will correspond to the `DTEMP` case control command if time assigned temperatures are added to the temperature set.

The only type of load that you can add to a temperature set is a temperature. You cannot add a temperature directly to the load container of an NX Nastran, MSC Nastran, or NX Multiphysics solution or subcase; instead, you must use a temperature set.

Note

You cannot add multiple temperature sets of the same type to a solution. For Nastran, you cannot include both an **Initial Temperature** type temperature set and a **Material Temperature** type temperature set in the same solution. For NX Multiphysics, the **Material Temperature** type temperature set is not available.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver
Simulation Navigator	Right-click **Solver Sets** → **New** → **Temperature Set**

Boundary Condition ID

What is it?

You can now insert the boundary condition ID into a boundary condition name by including the symbol `%%ID`. For example, if you create a thermal void with an ID of 6 and set the name to `Void – %%ID`, the name appears as `Void – 6`.

You can also pass the boundary condition ID as an argument of some built-in functions when you define boundary conditions using the expressions.

You can do this for any type of boundary condition that includes the boundary condition ID in its dialog box. Currently, these boundary conditions are:

- **Thermal Convecting Zone**
- **Thermal Stream**
- **Thermal Void**

Built-in functions with the boundary condition ID as an argument

In the following functions, “i” represents the boundary condition ID value.

Built-in function	Description
STI(i)	Returns the total absolute or relative inlet temperature of the stream i.
STO(i)	Returns the total absolute or relative outlet temperature of the stream i.
VT(i)	Returns the total absolute or relative temperature of the void i.
ZT(i)	Returns the total absolute or relative temperature of the convecting zone i.
SMO(i)	Returns the outlet mass flow of the stream i.
STMO(i)	Returns the outlet metal temperature of the stream i.
SP(i)	Returns the pressure of the stream i.
VP(i)	Returns the pressure of the void i.
ZP(i)	Returns the pressure of the convecting zone i.
SSV(i)	Returns the swirl velocity of the stream i.
VSV(i)	Returns the swirl velocity of the void i.
ZSV(i)	Returns the swirl velocity of the convecting zone i.
SA(i)	Returns the surface area of the stream i selection.
VA(i)	Returns the surface area of the void i selection.
ZA(i)	Returns the surface area of the convecting zone I selection.
MIX(i1,i2,...,i10)	Returns the temperature from mixing the flows of two or more streams, i1, i2, up to i10.
MMIX(i1,i2,...,i10)	Returns the mass flow from mixing the flows of two or more streams, i1, i2, up to i10.

Why should I use it?

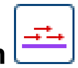
You can use the boundary condition ID to define interdependencies between boundary conditions; for example, to model flow recirculation of thermal streams, you define the outlet temperature of one thermal stream as a function of the input temperature of another stream.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Structural	Multi-Step Nonlinear
	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics or NX Thermal and Flow as the specified solver.

Command Finder	**Thermal Convecting Zone**  or **Thermal Stream**  or **Thermal Void** 
Simulation Navigator	Right-click the **Load** container node→**New Load**→**Thermal Convecting Zone** or **Thermal Stream** or **Thermal Void**
Location in dialog box	**Name** group→**Boundary Condition ID**

Temperature load enhancements

What is it?

You can now define a temperature load by importing temperature results into the Nastran and NX Multiphysics environments. You do this using a **Temperature — External Time Unassigned** type of **Temperature** load. You can import constant temperatures from Nastran, Abaqus, ANSYS, and NX Thermal results files.

In an NX Multiphysics environment, you can now do the following:

- Create temperature loads that are dependent on time. You create a **Temperature — Time Assigned** type of **Temperature** load to define temperatures that vary with time or with time and space. The temperature loads in any NX Multiphysics step must be either all time assigned or all not time assigned.

- Import time dependent temperature fields from an NX Thermal results file. You do this using a **Temperature — External Time Assigned** type of **Temperature** load.

Corresponding Nastran syntax

The new types of **Temperature** load correspond to different NX Nastran bulk data entries.

- If you select the **Temperature — Time Assigned** type, NX creates the `DTEMP` bulk data entry.

- If you select the **Temperature — External Time Assigned** type, NX creates the `DTEMPEX` bulk data entry.

- For all other types, NX creates the `TEMP` bulk data entry.

Defining initial temperatures and material temperatures

For NX Nastran, MSC Nastran, or NX Multiphysics, the **Temperature Load** command has been renamed to **Temperature**, as you can now use it to also define initial temperatures and material temperatures. You cannot add a **Temperature** load directly to the load container of an NX Nastran, MSC Nastran, or NX Multiphysics solution or subcase; instead, you must use a temperature set. The type that you select in the **Temperature Set** dialog box defines how the temperature will be used:

- An **Initial Temperature** type of temperature set defines the initial temperature distribution.

- A **Material Temperature** type of temperature set defines temperature-dependent material properties.

- A **Temperature Load** type of temperature set updates the temperature-dependent material properties, and defines the temperatures used for thermal loading.

For more information on the new usage of the **Temperature** load in Nastran, see Updated processes for defining temperatures in a Nastran analysis.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver Structural or Coupled as the specified analysis type
Command Finder	**Temperature** 

Updated processes for defining temperatures in a Nastran analysis

What is it?

The processes for defining temperatures in a Nastran analysis have been simplified for NX 10. You can now define all of the temperatures needed for studying thermal strains and stresses in a structural solution using the **Temperature** load and **Temperature Sets**.

Defining default temperatures

In previous releases, you used the following options in the **Solution** or **Solution Step** dialog boxes to define default temperatures:

- **Initial Temperatures Default**
- **Material Temperatures Default**
- **Temperature Load Default**
- **Default Initialization Temperature**

You now use the **Default** field in the **Temperature Set** dialog box to define your default temperatures. The **Type** option you select in the **Temperature Set** dialog box defines if the default will be an initial temperature, material temperature, or a temperature load.

Assigning temperatures to geometry

In previous releases, you used the following simulation objects and loads to assign temperatures to selected geometry:

- **Initial Temperatures** simulation object
- **Material Temperatures** simulation object
- **Temperature Load**

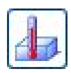
You now use a **Temperature** load each time you assign temperatures to selected geometry. The type of **Temperature Set** into which you add the **Temperature** load defines if the temperature will be an initial temperature, a material temperature, or a temperature load.

Applying results from a previous analysis

In previous releases, you used the **Temperature Pre-Load** option in the **Solution** or **Solution Step** dialog boxes to apply temperature results from a previous analysis.

You now use a **Temperature — External Time Unassigned** type of **Temperature** load to apply temperature results from a previous analysis.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver Structural or Coupled as the specified analysis type
Command Finder	**Temperature** 

Enforced motion load

What is it?

You can use the **Enforced Motion Load** command to create a known displacement to geometry or FE entities in a static or dynamic analysis.

- The **Components** type lets you define the displacement by the 6 degrees of freedom relative to a selected coordinate system.
- The **Magnitude and Direction** type lets you define the magnitude of the displacement in a single direction.
- The **Normal** type lets you define the constraint by specifying a displacement magnitude and a +/– direction that is normal to the selected geometry or element faces.

Using enforced motions in dynamic analysis

In a dynamic analysis, you can use the **Enforced Motion Load** command to also define a known velocity or acceleration.

In Nastran, you use the `SPCD` bulk data entry to define the enforced motion. You use the `TLOAD1` or `TLOAD2` bulk data entries to define a transient response excitation with a reference to the `SPCD` entries. You define the motion as a displacement, velocity, or acceleration with the `TYPE` field in the `TLOAD1` or `TLOAD2` bulk data entries.

In NX, you create an **Enforced Displacement**, **Enforced Velocity**, or **Enforced Acceleration** type of **Transient Excitation Load Set**. You add the **Enforced Motion Load** to the load set to define the type of motion used.

In Nastran, you use the `RLOAD1` or `RLOAD2` bulk data entries to define a frequency response excitation with a reference to the `SPCD` entries. You define the motion as a displacement, velocity, or acceleration with the `TYPE` field in the `RLOAD1` or `RLOAD2` bulk data entries.

In NX, you create an **Enforced Displacement**, **Enforced Velocity**, or **Enforced Acceleration** type of **Frequency Excitation Load Set**. You add the **Enforced Motion Load** to the load set to define the type of motion used.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver
Command finder	**Enforced Motion Load**

Pressure load enhancements

What is it?

For **Pressure** loads of the type **On Axisymmetric Elements (Normal to Edge) XY** and **On Axisymmetric Elements (Normal to Edge) ZX**, you can now define the pressure by specifying the total load to apply.

NX converts the total load to a pressure when it writes the input file:

Pressure = Total Force / Area of the 360° revolved edge

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver Axisymmetric Structural as the specified analysis type
Command Finder	**Pressure**

Edge load enhancements

What is it?

For edge loads of the type **Edge Load on Plane Strain and Stress Elements (Normal to Edge) XY** and **Edge Load on Plane Strain and Stress Elements (Normal to Edge) ZX**, you can now specify the total load to apply. With the **Constant** and **Spatial** distribution types, you define the load using pressure units, whereas with the new **Total** distribution type, the load is defined using force units.

NX converts the total load to a pressure when it writes the input file:

Pressure = Total Force / Element Edge Length x Element Thickness

Where do I find it?

Application	Advanced Simulation

Prerequisites	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver Axisymmetric Structural as the specified analysis type
Command Finder	**Edge Load**

Dynamic load time delay

What is it?

You can now create time delays for dynamic loads, using the **Dynamic Load Time Delay** modeling object. The same time delay can be applied to every excitation selected, or you can specify unique values for individual DOF components. The nodes affected can be individually selected, or chosen as a group.

You can define load time delay while creating a **Frequency Excitation**, **Transient Excitation**, or **Acoustic Source Load Set**. When defined with the load set, the delay can be created as a **Dynamic Load Time Delay** modeling object, or as a simple expression. If you use an expression, the delay is applied to all excitations in the load set.

Corresponding Nastran Syntax

The **Dynamic Load Time Delay** command corresponds to the NX Nastran `DELAY` bulk data entry.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver
Command Finder	**Modeling Objects**
Location in dialog box	**Type** list → **Dynamic Load Time Delay**

Dynamic load phase lead

What is it?

You can now create phase leads for dynamic loads, using the **Dynamic Load Phase Lead** modeling object. Apply the same angle of lead to every excitation selected, or specify unique values for individual DOF components. You can select the affected nodes individually or as a group.

You can define load phase lead while creating a **Frequency Excitation**, or an **Acoustic Source Load Set**. When defined with the load set, the delay can be created as a **Dynamic Load Phase Lead** modeling object, or as a simple expression. If you use an expression, the delay is applied to all excitations in the load set.

Corresponding Nastran Syntax

The **Dynamic Load Phase Lead** command corresponds to the NX Nastran `DPHASE` bulk data entry.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver
Command Finder	**Modeling Objects**
Location in dialog box	**Type** list → **Dynamic Load Phase Lead**

Darea Nodal Force and Moment load

What is it?

You can use the **Darea Nodal Force and Moment** load to create both forces and moments in which the values can vary at each node. You define the forces and moments by the six DOF components relative to the nodal displacement coordinate system.

By selecting the **Unique DOF Values** option, you can access the **Edit By Node** dialog box and enter unique values for each node.

Corresponding Nastran Syntax

The **Darea Nodal Force and Moment** command corresponds to the NX Nastran `DAREA` bulk data entry.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran or MSC Nastran as the specified solver
Command Finder	**Darea Nodal Force and Moment**

Acoustic source load

What is it?

Acoustic waves are longitudinal waves generated by the volumetric expansion or contraction of the medium (for example, air or water), caused by excitation of the system. You can use the new **Darea Acoustic Source** command and **Acoustic Source Load Set** to simulate a stationary acoustic source in the space that emits pulsating pressure waves. You can use this source for acoustic or vibro-acoustic analysis.

Examples of acoustic sources include:

- A small speaker in a small rigid box emitting sound at low frequency.
- An engine inside an anechoic chamber.
- An engine exhaust pipe outlet.

The sound waves emitted from an acoustic source form a uniform spherical envelope.

Using the **Darea Acoustic Source** command, you define the spatial distribution of the acoustic source by selecting nodes in a fluid mesh. To enable the acoustic source, enter **1** in the **DOF1** box. To disable the acoustic source, enter **0**.

You can use the **Unique DOF Values** option to enable or disable the acoustic source for each node independently. This way, you can study the contribution of each acoustic source independently, for an acoustic or vibro-acoustic system that has multiple acoustic sources.

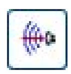
You must create the acoustic source in an **Acoustic Source Load Set**. This load set has an excitation value (real value only) that defines the `TP` field on the `ACSRCE` bulk data entry. For more information, see Acoustic Source Load Set.

Corresponding Nastran Syntax

The **Darea Acoustic Source** command corresponds to the `DAREA` bulk data entry.

The **Acoustic Source Load Set** command corresponds to the `ACSRCE` bulk data entry.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran or MSC Nastran as the specified solver. SOL 108 or SOL 111 as the specified solution type. **Acoustic** or **Vibro-Acoustic** as the specified analysis type. Mesh of Acoustic Fluid elements.
Command Finder	**Darea Acoustic Source** 

Bolt Pre-Load enhancements

What is it?

When defining **Bolt Pre-Load** for NX Nastran solutions 106, 129, and 601, as well as NX Multiphysics, you now define the bolt geometry from polygon bodies or 3D elements. You define the bolt plane from a point, a node, or a mesh point, and the bolt axis.

Force on 2D elements

In NX Multiphysics, you can also now create bolt pre-loads on 2D elements. You define the bolt geometry from polygon faces or 2D elements. You define the bolt plane from a point, a node, or a mesh point, and the bolt axis.

Corresponding Nastran Syntax

The changes in how the **Bolt Pre-Load** is defined correspond to a change in the option used on the Nastran `BOLT` bulk data entry. The **Bolt Pre-Load** for these solutions now uses the `ETYPE=3` option. In previous releases, the `ETYPE=2` option was used.

Where do I find it?

Application	Advanced Simulation
Command finder	**Bolt Pre-Load**

Abaqus axisymmetric contact analysis (10.0.1)

What is it?

This release includes expanded support for contact analyses in the Abaqus environment. You can now perform a number of different types of contact analyses in axisymmetric structural and thermal models. In previous releases, contact analyses in the Abaqus environment were limited to non-axisymmetric models.

The following table summarizes the commands that are now available for axisymmetric structural and thermal solutions:

Abaqus element	Description	Analysis types supported	Corresponding Abaqus keywords
Edge-to-Edge Contact	Defines contact conditions between selected polygon edges or element edges.	Axisymmetric structural	`*CONTACT PAIR` `*SURFACE`
Contact Interference	Defines time-dependent allowable interferences for contact pairs and elements. You can use **Contact Interference** to model interference fits (press fits) when there are large overclosures between the contacting surfaces in the initial configuration of the model. The defined contact interferences can result in stresses and strains as the solver resolves the overclosure between the contacting surfaces.	Axisymmetric structural	`*CONTACT PAIR` `*CONTACT INTERFERENCE`

Abaqus element	Description	Analysis types supported	Corresponding Abaqus keywords
Contact with Clearance 	Defines precise clearance or overclosure values for the nodes on the slave (target) surface in a contact pair. An overclosure value allows you to specify any initial penetration for the contacting surfaces.	Axisymmetric structural	`*CONTACT PAIR` `*CLEARANCE`
Surface-to-Surface Thermal Conductance 	Models conductive heat transfer between proximate or contacting surfaces. You can model the conductive heat transfer as a function of: • The clearance between the contacting surfaces. • The contact pressure at the interface between the contacting surfaces. • Both the clearance and the contact pressure.	Axisymmetric thermal	`*GAP CONDUCTANCE`

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver Axisymmetric Structural or Axisymmetric Thermal as the analysis type General Analysis or Heat Transfer as the solution type

ANSYS axisymmetric contact analysis (10.0.1)

What is it?

This release includes support for contact analyses in ANSYS axisymmetric structural and thermal solutions. In previous releases, contact was supported only in non-axisymmetric solutions.

Axisymmetric structural solutions

You can now use the **Structural Contact** command to analyze contact between the edges of planar elements in axisymmetric models. You use the **Structural Contact** and the **Region** dialog boxes to define contact and sliding between a 2D target edge and a deformable surface. When you export or solve the solution, the software creates the appropriate CONTA172 elements to define the deformable surface and TARGE169 elements to define the target surface.

Axisymmetric thermal solutions

You can now use the **Thermal Contact** command to define thermal contact conduction between contacting surfaces in axisymmetric models using the edges of planar elements. You use the **Thermal Contact** and the **Region** dialog boxes to define the contact between the edges, as well as select the edges, points, or nodes that define the contacting edges.

After you define the contacting surfaces, use the **CONTA174/172 Real Constants** dialog box to define the required thermal contact properties. You can define the following properties:

- **Thermal contact conductance** (`TCC`), which allows you to take into account the conductive heat transfer between the contact and target surfaces.

  Note

  Currently, NX supports only a constant **Thermal contact conductance** value. You cannot vary the **Thermal contact conductance** with temperature, pressure, or time.

- The **Stefan-Boltzmann constant** (`SBCT`), which you can use to model radiative heat transfer. If you do not specify a value for the **Stefan-Boltzmann constant**, the radiation effect is excluded from the thermal contact calculations.

- The **Radiation view factor** (`RDVF`), which you can use to define the radiation view factor for near-field radiation.

Where do I find it?

Structural Contact

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Axisymmetric Structural as the analysis type Linear Statics or Nonlinear Statics as the solution type
Command Finder	**Structural Contact** 

Thermal Contact

Application	Advanced Simulation

Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Axisymmetric Thermal as the analysis type Steady State Thermal as the solution type
Command Finder	**Thermal Contact**

Constraint equation support in Abaqus and ANSYS thermal solutions (10.0.1)

What is it?

You can now create constraint equations between the temperature degrees-of-freedom of thermal elements in both the Abaqus and ANSYS thermal environments. In previous releases, you could only create constraint equations in the Abaqus and ANSYS structural environments.

Constraint equations for Abaqus thermal solutions

In the Abaqus thermal environment, use the new **Manual Coupling** command to define the independent and dependent coupling parameters between selected nodes. When you export or solve your model, NX creates the appropriate `*EQUATION` commands in your Abaqus input file.

Constraint equations for ANSYS thermal solutions

In the ANSYS thermal environment, the **Manual Thermal Coupling** command has been enhanced to allow you to create constraint equations as well as define sets of coupled degrees-of-freedom. Use the new **CE** option in the **Type** list of the **Manual Thermal Coupling** dialog box to define coupling between the temperature degrees-of-freedom within the model. When you export or solve your model, the software creates the appropriate `CE` keywords in your ANSYS input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus or ANSYS as the specified solver Thermal as the analysis type
Command Finder	**Manual Coupling** (Abaqus) **Manual Thermal Coupling** (ANSYS)

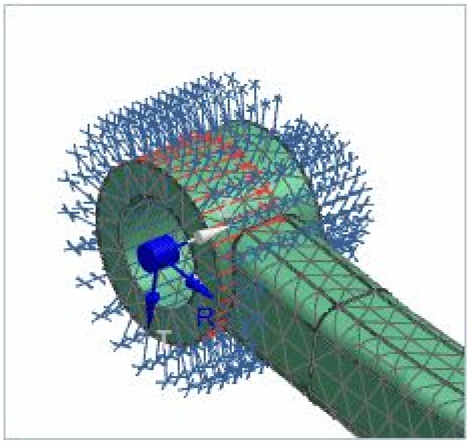
Displaying boundary conditions and associated geometry

What is it?

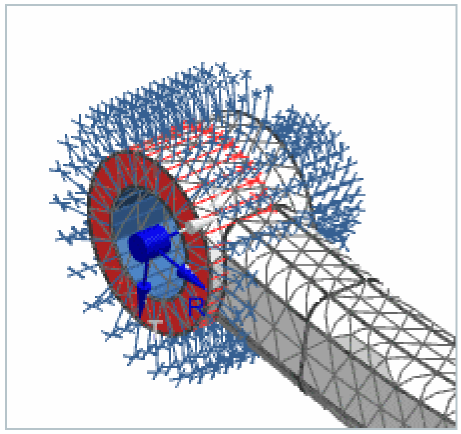
This release includes new commands and options that let you control the display of boundary conditions and geometry associated with the boundary conditions.

- Use the **BC Selection Display** command to highlight geometry that is associated with selected boundary conditions. The geometry is displayed with the same color as the boundary condition symbols and labels. This display mode is an alternative to the display that is specified by the **Color Basis** option and in the **Model Display Options** dialog box.

  The following image shows the view of pinned and user-defined constraints on a face when the **BC Selection Display** command is not selected, and when it is.



Standard view



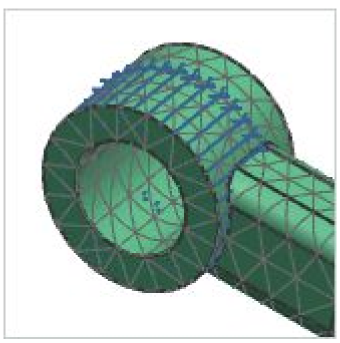
BC Selection Display selected

- Use the **BC Selection Display Options** command to select which boundary conditions are displayed. You can also control the display colors for excluded and overlapping boundary conditions.

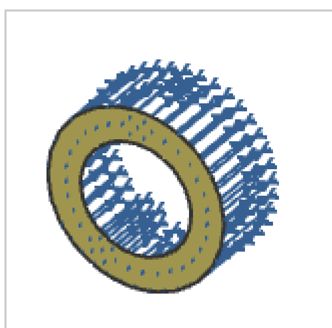
When you right-click a boundary condition name, two new options provide additional refinements to the display.

- **Show Only** displays only the selected boundary conditions and their associated geometry.

  The following image shows the views of a user-defined constraint on a face when the **Show Only** option is not selected, and when it is.



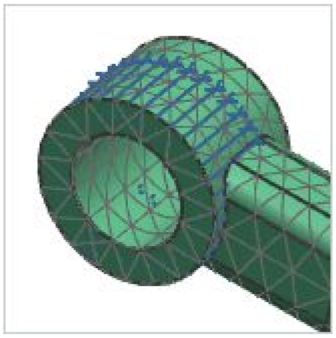
Standard view



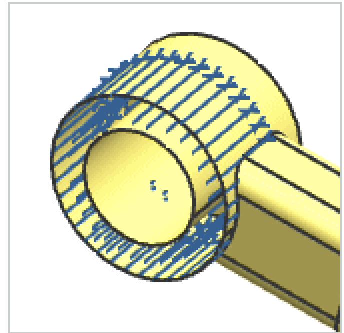
Show Only selected

- **Show Reverse** displays everything except the selected boundary conditions and their associated geometry. This option lets you determine whether or not you have applied a boundary condition to a particular geometry item.

The following image shows the views of a user-defined constraint on a face when the **Show Reverse** option is not selected, and when it is.



Standard view



Show Reverse selected

Where do I find it?

BC Selection Display and **BC Selection Display Options**

Application	Advanced Simulation
Prerequisite	Boundary conditions applied to the model
Command Finder	**BC Selection Display** or **BC Selection Display Options**

Show Only and **Show Reverse**

Application	Advanced Simulation
Prerequisite	Boundary conditions applied to the model
Simulation Navigator	Right-click the boundary condition→**Show Only** or **Show Reverse**

User interface changes for defining boundary conditions

What is it?

The dialog boxes that you use to specify a boundary condition have changed in NX 10.

- In earlier versions of NX, to specify the magnitude of a boundary condition, you first select either **Expression** or **Field** from the drop-down list in the **Magnitude** group. If you select **Field**, you are then presented with a second drop-down list. From this list, you select how you want to define the field. For many boundary conditions in NX 10, you specify the boundary condition from a single entry box regardless of whether you want to define it with an expression or a field.

- In earlier versions of NX, the magnitude and distribution of a boundary condition are defined separately. The magnitude is defined in the **Magnitude** group, and the distribution is defined in the **Distribution** group. For many boundary conditions in NX 10, both the magnitude and distribution of the boundary condition are defined by a single expression or field entry in the **Magnitude** group.

- In earlier versions of NX, the **Scale Factor** box appears in the **Magnitude** group once you select **Field** from the drop-down list. In NX 10, the **Scale Factor** box does not appear when you define the boundary condition with a field. Instead, an edit button appears in the entry box. If you click the edit button, an option list appears. If you select **Scale Factor** from the list, the new **Field Scale Factor** dialog box opens from which you can enter the scale factor.

Where do I find it?

Entering a scale factor	
Application	Advanced Simulation
Prerequisite	Define the magnitude and distribution of a boundary condition with a field
Location in dialog box	[Boundary condition] dialog box → **Magnitude** group → edit  → list → **Scale Factor** → **Field Scale Factor** dialog box → **Table Field** box

Solutions

Updating a solution from a condition sequence

What is it?

You can now modify a condition sequence (such as add, modify, and remove time points), and then use the new **Update from Condition Sequence** command to update the solution that was created by that condition sequence. In the previous release, you could not edit a condition sequence and update the steps in the solution that was created from the condition sequence.

The **Update from Condition Sequence** command makes corresponding changes to the time steps in the solution.

- If you add new time points to the condition sequence, the update command adds new condition sequence-associated time steps to the solution.
- If you change the time values in the condition sequence, the update command updates the solution step with the new time values.
- If you remove time points from the condition sequence, the update command deletes the solution steps associated with those time points.

Updating a solution with a different condition sequence

For a solution that was created from a condition sequence, you can also use a different condition sequence to update the solution. For example, suppose you use condition sequence **CS-A** to create Solution 123. Later, you create condition sequence **CS-B**. You can use the **Update from Condition Sequence** command to update Solution 123 with condition sequence **CS-B**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the work part and displayed part A solution created from a condition sequence, using the **New Solution from Condition Sequence** command
Simulation Navigator	Right-click the solution→**Update from Condition Sequence**

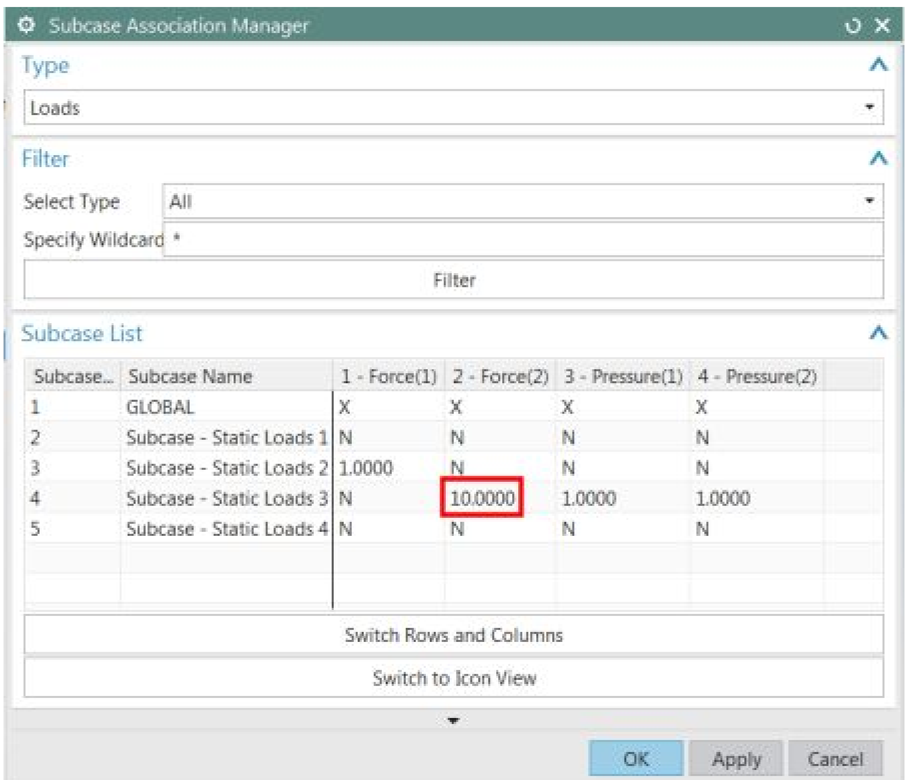
Scaling loads using the Subcase/Step Association Manager

What is it?

You can now scale static loads directly in the **Subcase/Step Association Manager** in the following environments:

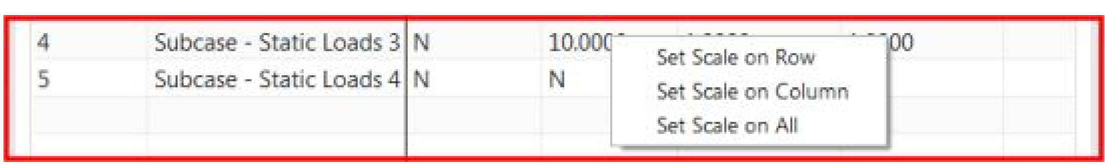
- NX Multiphysics Structural and Coupled environments
- NX Nastran Structural (solution 101 only)

A new **Switch to Scaling View** button allows you to add a real number as a scale factor for individual loads and for defined load sets. A scale factor defined for a load set applies to all static loads in that set. The following graphic shows the scaling view; the highlighted value shows a scale factor of 10 applied to a force load in a subcase.



After you specify a scale factor for a load-step association, you can right-click that scale factor and choose an option that applies that scale factor to other loads.

- To apply it to that same load in all steps, choose **Set Scale on Column**.
- To apply it to all loads in one step, choose **Set Scale on Row**.
- To apply it to all loads in all steps, choose **Set Scale on All**.



The scale factors appear in the Nastran `LOAD` bulk data entries.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran (SOL 101 only) or NX Multiphysics as the specified solver A Simulation file as the work and displayed part At least one active solution that contains multiple steps or subcases
Simulation Navigator	Right-click a selected solution→**Step Manager** or **Subcase Manager** (Nastran)
Location in dialog box	**Switch to Scaling View** button

Making copies of subcases and steps

What is it?

You can now make copies of subcases or steps using the **Clone Multiple** command.

When you use this command:

- The cloned copies include the loads and constraints from the original subcase or step.
- The name of each cloned copy includes the original entity plus a numerical suffix.
- The last copy created becomes the new active subcase or step.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	A Simulation file as the work part and displayed part
Simulation Navigator	Right-click a subcase or step → **Clone Multiple**

Reordering steps

What is it?

You can reorder the steps or subcases in a solution by using drag and drop in the **Reorder** dialog box. You can reorder multiple steps by dragging them at the same time.

Note

Reordering your steps will not change their end time. You can change the end time of steps using the **End Time** field in the **Solution Step** dialog box.

When NX can infer the boundary condition evaluation time or the end time, it displays it in the **Time/End Time** column. When NX can determine a boundary condition evaluation frequency or a forcing frequency, it displays it in the **Frequency** column.

When you create additional steps in your solution, you can control the order in which the new steps are placed. By default, NX creates the new step in the last position. To place your new step before an existing step, select the existing step from the **Relative Position** field in the **Solution Step** dialog box.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click a solution → **Reorder Steps** or **Reorder Subcases**

Editing boundary condition-step association using a spreadsheet

What is it?

You can now export boundary condition-step associations to a spreadsheet, make changes in the spreadsheet, and then import the associations back into NX. In previous versions, you could export the associations, but you could not import them back into NX.

	A	B	C	D	E
1	Subcase Name	Force(1)	Force(2)	Pressure(1)	Pressure(2)
2	GLOBAL	X	X	X	X
3	Subcase - Static Loads 1	N	N	N	N
4	Subcase - Static Loads 2	20.0000	N	N	N
5	Subcase - Static Loads 3	N	10.0000	50.0000	1.0000
6	Subcase - Static Loads 4	N	N	N	N

With this release, use these steps:

1. To edit your data in a spreadsheet, in the **Step/Subcase Manager** dialog box, right-click in an empty area of the **Step/Subcase List** table and choose **Edit in Spreadsheet**.

2. Make your changes and save the spreadsheet as a comma-separated values (CSV) file.

3. To import your edited file, in the **Step/Subcase Manager** dialog box, right-click in an empty area of the **Step/Subcase List** table and choose **Import from CSV**.

This release also adds the **Export to CSV** command, which lets you export the association data from NX to a file in the comma-separated values format.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	A Simulation file as the work and displayed part At least one active solution that contains multiple steps or subcases
Simulation Navigator	Right-click a selected solution→**Step Manager** or **Subcase Manager** (Nastran)

Generating loads and subcases from a recipe

What is it?

You can now import load data from an external source and use it to generate solutions, subcases, and loads. Use the **Load Recipes Manager** command to define a load recipe that specifies the type of load data and how it should be mapped to nodes and faces in a FEM. You can then use the **New Solution from Load Recipe** command to create a new solution.

The typical use of a load recipe is to apply loads to an acoustics simulation model. Using a load recipe, you can apply function data to the entities of the model.

Load Recipe dialog box

The load recipe defines how the boundary condition and subcases are created with the data from data sources. In the **Load Recipe** dialog box, you can define the following:

Data Sources

You can add multiple data sources in the load recipe using the **Add** button. For the selected data source, a format handler is available, where you can specify how the data from the data sources is interpreted. Only the universal (.unv) file type is supported.

Load Conditions

You can specify the load conditions that are used for the creation of subcases by enabling or disabling a load condition.

Waterfall

You can get an overview of the available tracking values.

Mapping

You can map a function from the data sources to a node or face of the model. You can select a load from the available load types.

You can map the loads to the target using these methods:

- **Automatic Entity (AE)** — Maps a function from the data sources to the defined entity label and the component.

- **Manual Function (MF)** — Maps a function with the specified function name from the data sources to the one on the model, even though it may not have the correct entity and DOF annotations.

- **Manual Entity (ME)** — Maps a function from the data sources to the specified entity and the component, but ignores the entity label of the target.

Load Recipes Manager

The **Load Recipes Manager** is a collector of load recipes that you can use to create, delete, or edit the load recipes.

- You can attach the function from the data sources to the node or to the face by using various tabs on the **Load Recipe** dialog box.

- You can define the recipe with the data from an external source and apply it to the model.

- You can create subcases and loads based on the available data in the external source and assign loads to the subcases.

Creating solutions from a load recipe

You can create a solution from a load recipe using the **New Solution from Load Recipe** command. The **Solution** dialog box lists only those analysis types and solution types that the load recipe supports. NX generates the loads and places them in the boundary condition containers, depending on the load type defined in the load recipes. For multiple subcases, you can create the folders in the boundary condition container. Each of these folders contains all the loads necessary for creating one subcase.

Update solution from a load recipe

You can update a solution from a load recipe using the **New Solution from Load Recipe** command. It updates the subcases with the related boundary conditions and table fields, depending on options that you select in the **Solution** dialog box.

Where do I find it?

Importing, editing, or creating load recipes

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and displayed part
Command Finder	**Load Recipes** 
Menu	**Insert→Load Recipes**

Creating a new solution from a load recipe

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and displayed part
Menu	**Insert→New Solution from Load Recipes**
Simulation Navigator	Right-click the **Simulation** node→**New Solution from Load Recipes**

Nastran support enhancements

NX Nastran acoustics analysis support in NX

What is it?

This release includes significant enhancements to support NX Nastran acoustic and vibro-acoustic analysis.

Acoustic and Vibro-Acoustic solutions

When you create a new NX Nastran solution, two new analysis types are available:

- **Acoustic** — only acoustic elements (3D) are available in this environment.
- **Vibro-Acoustic** — both acoustic elements (3D) and structural elements are available in this environment.

Supported NX Nastran solutions

The following solution types are supported with both acoustic and vibro-acoustic solutions:

- SOL 103 – Real Eigenvalues
- SOL 107 – Direct Complex Eigenvalues
- SOL 108 – Direct Frequency Response
- SOL 110 – Modal Complex Eigenvalues
- SOL 111 – Modal Frequency Response

Supported acoustics capabilities

Acoustic Absorber

You can create an **Acoustic Absorber** simulation object in acoustic or vibro-acoustic solutions. An acoustic absorber is applied on free faces of the fluid elements. It allows you to define an acoustic surface impedance (or its reciprocal admittance). The **Acoustic Absorber** corresponds to the Nastran `CAABSF` and `PAABSF` bulk data entries.

Panel

You can create a **Panel** simulation object in vibro-acoustic solutions. A panel contains 2D structural elements. It corresponds to the `PANEL` bulk data entry. It can be used to compute contributions of structural panels to acoustic pressure results.

Fluid-Structure Interface Modeling Parameters

For a vibro-acoustic solution, to specify the acoustic and structural elements that need to be coupled, create a **Fluid-Structure Interface Modeling Parameters** modeling object, which corresponds to the `ACMODL` bulk data entry. You can define the wetted surface by selecting the acoustic and structural elements to be coupled. You can create multiple modeling objects, but can select only one modeling object per solution (on the **Bulk Data** tab of the **Solution** dialog box).

Fluid-Structure Interaction Control Parameters

For a vibro-acoustic solution, to define parameters for the fluid-structure interaction, create a **Fluid-Structure Interaction Control Parameters** modeling object, which corresponds to the `FLSTCNT` case control command.

Modal Damping

For a vibro-acoustic SOL 111 solution, you can define modal damping separately for the structural and the acoustic model. You can define modal damping groups on the:

- **Case Control** tab of the **Solution** dialog box

- **Solution Step** dialog box for each subcase

You can define uniform structural damping and uniform fluid damping in the damping parameter groups on the **Parameters** tab of the **Solution** dialog box. These options correspond to the `KDAMP,KDAMPFL` and `G,GFL` parameters.

Modal Contributions for Structural, Acoustic and Vibro-Acoustic analysis types

You can define the modal contribution parameters in structural, acoustic, or vibro-acoustic solutions, in their **Output Requests**. The **Modal Contribution** output request corresponds to the `MODCON` case control command.

In a vibro-acoustic solution, you can request modal contributions separately for the structural modes and the fluid modes.

You can request SORT1 and SORT2 output formats. However, only contributions in SORT2 format are supported for display in the NX **XY Function Navigator**. The SORT1 output can be in the form of PRINT and/or PUNCH. NX Nastran will then write the results to the f06 or to the PCH file, respectively.

Panel Contributions for Acoustic and Vibro-Acoustic analysis types

You can request the panel contributions in acoustic or vibro-acoustic solutions in their **Output Requests**. These are the contributions from structural panels to pressure results on acoustic fluid nodes for the solution. The **Panel Contribution** output request corresponds to the `PANCON` case control command.

Where do I find it?

Creating an acoustic or vibro-acoustic analysis

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and displayed part
Menu	**Insert** →**Solution**
Location in dialog box	**Solution** dialog box→ **Analysis Type** list→select **Acoustic** or **Vibro-Acoustic**

Creating an acoustic absorber

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and displayed part An active Acoustic or Vibro-Acoustic solution
Simulation Navigator	Right-click the **Simulation Object** node→**New Simulation Object**→**Acoustic Absorber**
Command Finder	**Acoustic Absorber** 

Creating a panel

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and the displayed part An active Acoustic or Vibro-Acoustic solution
Simulation Navigator	Right-click the **Simulation Object** node→**New Simulation Object**→**Panel**
Command Finder	**Panel**

Creating a fluid-structure interface

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and displayed part An active or Vibro-Acoustic solution
Menu	**Insert**→**Modeling Objects**→**Fluid-Structure Interaction Modeling Parameters** or **Fluid-Structure Interface Modeling Parameters**
Location in dialog box	**Modeling Objects Manager** dialog box→**Type** list→**Fluid-Structure Interface Modeling Parameters** or **Fluid-Structure Interaction Control Parameters**

Analyzing vibro-acoustic contribution data

What is it?

After you solve a vibro-acoustic model, you can analyze the panel contribution data and modal contribution data using the **XY Function Navigator**.

You can plot the result curves for a particular subcase, type, or degree of freedom for all contribution data.

Analyzing contribution data

You can request the following types of contribution results in the Nastran solution.

Panel Contributions

Panel contribution curves express the contribution of a given vibrating structural panel (the sum of all its nodes) to the pressure response at a given acoustic node. Panels must be part of the wetted surface. You can request this contribution using the **Panel Contribution** output request for the solution by selecting the **Panels for Contributions** option.

Structural Mode Contributions

Structural mode contributions express the contribution of a given structural mode to the response at a given node and degree of freedom. You can request this contribution using the **Modal Contribution** output request for the solution by selecting the **Structural Modes for Contributions** option.

Acoustic Mode Contributions

Acoustic mode contributions express the contribution of a given acoustic mode to the response at a given node and degree of freedom. You can request this contribution using the **Modal Contribution** output request for the solution by selecting the **Fluid Modes for Contributions** option.

The OP2 result file from the NX Nastran solution will contain the requested contribution data.

XY plotting from OP2 SORT2 results

In NX, you can analyze the NX Nastran SORT2 results using the **XY Function Navigator**. The results are grouped by response node, response type, degree of freedom, and subcase.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A loaded Nastran OP2 SORT2 results file in the **XY Function Navigator**
XY Function Navigator	Right-click the **Nastran OP2 Files- Sort2 Results**→**Open** or **Open Directory**
Graphics window	Right-click the desired contribution curve and click **XY Plot**

Generating loads and subcases from a recipe

What is it?

You can now import load data from an external source and use it to generate solutions, subcases, and loads. Use the **Load Recipes Manager** command to define a load recipe that specifies the type of load data and how it should be mapped to nodes and faces in a FEM. You can then use the **New Solution from Load Recipe** command to create a new solution.

The typical use of a load recipe is to apply loads to an acoustics simulation model. Using a load recipe, you can apply function data to the entities of the model.

Load Recipe dialog box

The load recipe defines how the boundary condition and subcases are created with the data from data sources. In the **Load Recipe** dialog box, you can define the following:

Data Sources

You can add multiple data sources in the load recipe using the **Add** button. For the selected data source, a format handler is available, where you can specify how the data from the data sources is interpreted. Only the universal (.unv) file type is supported.

Load Conditions

You can specify the load conditions that are used for the creation of subcases by enabling or disabling a load condition.

Waterfall

You can get an overview of the available tracking values.

Mapping

You can map a function from the data sources to a node or face of the model. You can select a load from the available load types.

You can map the loads to the target using these methods:

- **Automatic Entity (AE)** — Maps a function from the data sources to the defined entity label and the component.

- **Manual Function (MF)** — Maps a function with the specified function name from the data sources to the one on the model, even though it may not have the correct entity and DOF annotations.

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- You can attach the function from the data sources to the node or to the face by using various tabs on the **Load Recipe** dialog box.

- You can define the recipe with the data from an external source and apply it to the model.

- You can create subcases and loads based on the available data in the external source and assign loads to the subcases.

Creating solutions from a load recipe

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Update solution from a load recipe

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Where do I find it?

Importing, editing, or creating load recipes

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and displayed part

Command Finder	**Load Recipes**
Menu	**Insert→Load Recipes**

Creating a new solution from a load recipe

Application	Advanced Simulation
Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and displayed part
Menu	**Insert→New Solution from Load Recipes**
Simulation Navigator	Right-click the **Simulation** node→**New Solution from Load Recipes**

PBEAR physical property table enhancements

What is it?

This release includes a number of enhancements for the **PBEAR** physical property table. In NX Nastran, you use the `PBEAR` bulk data entry to define the stiffness and damping matrices for bearing connections that you define with the `CBEAR` bulk data entry for rotor dynamics analyses.

You can use new options in the **PBEAR** dialog box to do the following:

- Define axial stiffness and viscous damping terms in the translation direction, and radial stiffness and viscous damping terms in the direction of rotation.
- Specify whether the components of the stiffness and viscous damping matrices are constant or vary as a function of speed, speed and relative displacement, or speed and relative force.
- Specify composite relative displacements and forces.
- Set nominal values for stiffness and viscous damping.

Support for new stiffness and damping terms

You now have more flexibility in terms of how you define the stiffness and damping terms for `CBEAR` elements. In previous releases, you could define the translational stiffness and viscous damping for `CBEAR` elements in the plane normal to the rotor's axis. Now, you can define the following:

- Axial stiffness and viscous damping terms in the translation direction.
- Radial stiffness and damping terms in the direction of rotation.
- The stiffness and damping of the cross coupling axial translational direction with other translational directions.

These enhancements allow you to create bearings that have combined radial and axial dependence.

Defining stiffness and damping components as constant or varying values

You can use the new **Field Independent Domain** option in the **PBEAR** dialog box to define stiffness and viscous damping matrices as a constant value or as a function of the following:

- Rotor speed.
- Rotor speed and relative displacement.
- Rotor speed and relative force.

Defining composite relative displacements and forces

Composite relative displacements are linear combinations of radial and axial relative displacements. You can now use the options on the new **Composite Coefficient** tab to define linear combinations of axial and radial relative displacements or axial and radial relative forces when the stiffness or viscous damping matrices vary with respect to rotor speed and relative displacement or relative force.

- For bearing properties that are speed and displacement-dependent, the software uses the composite relative displacements to look up values for bearing stiffness and bearing viscous damping.
- For bearing properties that are speed and force-dependent, the software uses the composite relative forces to look up values for bearing stiffness and bearing viscous damping.

Defining nominal values for stiffness or viscous damping

On the **Stiffness Matrix** and **Viscous Damping Matrix** tabs in the **PBEAR** dialog box, you can use the new **Nominal Relative Displacement** options to define the relative displacement or force that NX Nastran uses to compute the bearing stiffness or viscous damping.

Note

NX Nastran uses the specified nominal values only if the stiffness or damping matrices are not defined as a constant value.

Where do I find it?

CBEAR elements

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part NX Nastran version 10.0 as the specified solver
Command Finder	**1D Mesh**
Location in dialog box	**Type→CBEAR**

PBEAR dialog box

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part NX Nastran version 10.0 as the specified solver
Command Finder	**Physical Properties**

Location in dialog box	**Type→PBEAR**

Updated processes for defining temperatures in a Nastran analysis

What is it?

The processes for defining temperatures in a Nastran analysis have been simplified for NX 10. You can now define all of the temperatures needed for studying thermal strains and stresses in a structural solution using the **Temperature** load and **Temperature Sets**.

Defining default temperatures

In previous releases, you used the following options in the **Solution** or **Solution Step** dialog boxes to define default temperatures:

- **Initial Temperatures Default**
- **Material Temperatures Default**
- **Temperature Load Default**
- **Default Initialization Temperature**

You now use the **Default** field in the **Temperature Set** dialog box to define your default temperatures. The **Type** option you select in the **Temperature Set** dialog box defines if the default will be an initial temperature, material temperature, or a temperature load.

Assigning temperatures to geometry

In previous releases, you used the following simulation objects and loads to assign temperatures to selected geometry:

- **Initial Temperatures** simulation object
- **Material Temperatures** simulation object
- **Temperature Load**

You now use a **Temperature** load each time you assign temperatures to selected geometry. The type of **Temperature Set** into which you add the **Temperature** load defines if the temperature will be an initial temperature, a material temperature, or a temperature load.

Applying results from a previous analysis

In previous releases, you used the **Temperature Pre-Load** option in the **Solution** or **Solution Step** dialog boxes to apply temperature results from a previous analysis.

You now use a **Temperature — External Time Unassigned** type of **Temperature** load to apply temperature results from a previous analysis.

Where do I find it?

Application	Advanced Simulation

Prerequisites	NX Nastran, MSC Nastran, or NX Multiphysics as the specified solver Structural or Coupled as the specified analysis type
Command Finder	**Temperature**

Improved support for residual vector results

What is it?

This release includes improved support for the calculation of residual vectors.

- NX now supports the import of the `RESVEC` case control command from both NX Nastran and MSC Nastran input files. NX can import the `RESVEC` command from both .dat and .op2 files.

- NX now supports the `NODAMP` describer for the `RESVEC` case control command. If you do not want to compute residual vectors for the degree-of-freedom in which viscous damping is defined on `CBUSH`/, `CDAMP`/, and `CVISC` elements, select the new **NODAMP** option in the **Viscous Damping** list on the **Residual Vector** tab. In previous releases, NX only supported the `DAMPLOD` describer.

For more information on residual vectors, see the *NX Nastran Basic Dynamics User's Guide*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part NX Nastran or MSC Nastran as the specified solver
Command Finder	**Modeling Objects**
Location in Dialog Box	**Type** list→ **Structural Output Requests**→**Residual Vector** tab

Response Simulation support for composite elements

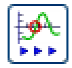
What is it?

You can now include composite elements in a Response Simulation analysis. For Random type of events, you can now generate RMS and LCR stress contour results when the dynamic solution results file contains ply stress and/or homogeneous stress modes. In previous releases, Response Simulation did not support composite elements.

In the **Evaluate RMS Results** and **Evaluate LCR Results** dialog boxes, you can select **Stress** as the **Result Type**. The composite elements in the model are evaluated with respect to the ply result coordinate system.

Where do I find it?

Application	Advanced Simulation

Prerequisites	NX Nastran as the specified solver A Simulation file as the work part and displayed part An active Response Simulation solution process
Command Finder	**Evaluate RMS Results**  **Evaluate LCR Results** 

Import improvements for models with material orientation vectors

What is it?

Larger finite element models can contain thousands of 3D elements, each with their own material orientation vector. When you create a Nastran input file from that model, the thousands of 3D elements become thousands of `PSOLID` bulk data entries, each possibly referencing a separate `CORDM` coordinate system bulk data entry. When you import an input file like this into NX, NX tries to create separate mesh collectors in the **Simulation Navigator** for each of these coordinate systems and `PSOLID` entries. This process of creating thousands of separate mesh collectors can seriously degrade the import performance.

You can use the **Import 3D MOVs as field data** option in the **Import Simulation** dialog box to automatically create spatial fields to represent the individual material orientation vectors. Importing 3D material orientation vectors as spatial fields can significantly improve import performance. However, importing the material orientation vectors as spatial fields can result in a loss of detail.

This release includes a new **Number of PSOLID properties for spatial field creation** customer default. Use this default to specify the threshold number of `PSOLID` properties that a solver input file must contain for NX to automatically create spatial fields to represent the material orientation vectors for 3D elements. NX creates the spatial fields when the number of `PSOLID` properties in the Nastran input file exceed this threshold value.

Where do I find it?

Import 3D MOVs as field data option

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part NX Nastran as the specified solver **Structural** as the specified analysis type
Menu	**File→Import→Simulation**
Location in dialog box	**Data Compaction Options** group

Number of PSOLID properties for spatial field creation customer default

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Nastran→General** tab

Updates to Geometry Check options

What is it?

You can now use the **Geometry Check Options** modeling object to run optional quality checks on Nastran plane strain, plane stress, and axisymmetric elements. You can now have Nastran perform the following optional checks when you solve your model:

- Minimum and maximum interior angle (`TRX_IAMN, TRX_IAMX, QDX_IAMN, QDX_IAMX`)
- Aspect ratio (`TRX_AR, QDX_AR`)
- Skew (`QDX_SKEW`)
- Taper (`QDX_TAPR`)
- Edge point length ratio (`TRX_EPLR, QDX_EPLR`)

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part NX Nastran or MSC Nastran as the specified solver
Command Finder	**Modeling Objects**
Location in Dialog Box	**Type** list→**Geometry Check Options**

Import now available for NX Nastran SOL 401 solutions

What is it?

You can now import NX Nastran SOL 401 input files into Advanced Simulation. For SOL 401 input files, select **NX MULTIPHYSICS** as the solver in the **Import** dialog box. You can import SOL 401 solutions from both .dat and .op2 files. If you select the **Create new solution** option in the **Import Simulation** dialog box, NX imports the data into a new solution with NX Multiphysics as the specified solver and the **Analysis Type** set to **Structural**.

Additionally, you can also use the **Append Merge** option in the **Solver Deck Append** dialog box to import, append, and merge data from a SOL 401 input file, which is the source file, into an NX Multiphysics FEM or Simulation file, which is the destination file.

Importing NX Multiphysics solutions that were previously exported from NX

Currently, you cannot use a **Round Trip Parameters** modeling object to preserve the order and format of bulk data that you import from a SOL 401 solution for later export. However, if you export an NX Multiphysics solution from NX and later import that same solution back into NX, the two solutions should be equivalent. A solve on the imported solution should generate the same results as a solve on the original solution.

Note

Some minor differences may exist between the two solutions. For example, how a solution appears in the **Simulation Navigator** may be slightly different. For example, loads and boundary conditions could be listed within a specific **Step** instead of with the global solution.

The following are not supported for re-import into NX following a previous export:

- The setting for the **Ignore Material Temperature Dependence** option on **Solution Control** tab in the **Solution** dialog box.

- The **Augmented Time Step** modeling object dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file as the displayed part and the work part NX Multiphysics as the specified solver
Menu	**File→Import→Simulation**

Import improvements for Nastran

What is it?

This release includes several new options on the **Import Simulation** dialog box. These new options provide you with additional control over how NX imports data from your Nastran input file.

New data checking options

Use the new **Data Checking** option in the **Import Simulation** dialog box to control the degree to which NX evaluates the validity of the file you are importing.

- Select **Minimal** to perform the lowest level of mesh validity checking. With this option, NX assumes that the input file is ready to solve, and NX only performs minor checks during the import. The **Minimal** option offers the fastest import performance.

- Select **Nominal** to have NX check for basic issues with the input file. For example, NX checks to ensure that nodes and elements that are referenced by loads are included in the file. The **Nominal** option offers a balance between file validity checking and import performance.

- Select **Extended** to perform the highest level of mesh validity checking. NX checks for the existence of each node on every element that is included in the input file. The **Extended** option can increase the processing time for importing an input file, particularly in larger models. In addition, NX checks each element for duplicate nodes.

  Note

  If you select any of the **Selective Import** options, NX performs **Extended** data checking.

New option for importing multi-point constraints

Use the new **Import MPCs as Multi-MPCs** option to improve the import performance for models that contain a large number (greater than 10,000) of multi-point constraints. When you select this option, NX imports the individual MPCs as a single **Multi-MPC** type of **Manual Coupling** constraint instead of numerous individual **MPC** type constraints. This option offers faster import times for a large model at the expense of a more detailed representation of your input file in NX.

This release also includes a new **Number of MPCs for Multi-MPC Creation** customer default. You can use this default to specify the threshold number of multi-point constraints that a solver input file must contain before NX imports them as **Multi-MPC** couplings instead of individual **MPC** couplings.

Where do I find it?

Import Simulation dialog box

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file as the displayed part and the work part NX Nastran or MSC Nastran as the specified solver
Menu	**File→Import→Simulation**

Number of MPCs for Multi-MPC Creation customer default

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Nastran→General** tab

Export improvements for Nastran

What is it?

This release includes several new options on the **Export Simulation** dialog box. These new options provide you with additional control over how NX exports data from your NX to your Nastran input file.

Control over exported temperature unit system

In this release, you can now control the units used for temperatures in your model during export. This gives you the flexibility to define temperatures in a different unit system, such as Kelvin, than the rest of your model, which may be in metric or British units. In previous releases, you could only specify a different unit system for temperatures during import.

In the **Export Simulation** dialog box, use the new **Specify temperature units** list to select the unit system to use for temperatures in the Nastran input file that you export. If you select a different unit system for temperatures, the software converts units for temperature values in your model during the export. These values include, for example:

- **Temperature** and **Radiation** loads
- **Initial Temperature** and **Material Temperature** simulation objects
- **Thermal Constraints** and **Convection** constraints

- Material properties that have temperature units, such as values for the **Thermal Expansion Coefficient** and **Phase Change Temperature**

Special export processing for input files that contain large ID values

NX now automatically detects large ID values within your Nastran input file and performs special processing during export. In previous releases, NX could export input files that contained large IDs, however, if the bulk data entry associated with the large ID was written using the small field format, then the export operation aborted.

- In Nastran, an ID value is defined as large when it requires more than eight digits, namely an ID greater than 99999999.

- In NX, node and element ID values cannot exceed 2^31-1 or 2147483647. However, you can use options in the **Formatting Options** group of the **Export Simulation** dialog box to offset these ID values by up to this same maximum limit.

Therefore, the largest ID value that NX can export for a Nastran input file is 2*(2^31-1), or 4294967294.

In NX 10, when you export an NX Nastran input file that contains large IDs, NX now does the following:

- Automatically exports the bulk data entry using the large field format.

  Note

  If you use **Solver Syntax Preview** to preview the format of a bulk data entry with a large ID, NX also uses the large field format to preview the entry.

- Writes out the `NASTRAN SYSTEM(525)=1` system cell in the Nastran section of the input file.

- Issues a warning or error message, as appropriate, during the export of the following types of solutions:

  o Warning message: SOL 101 Superelement, SOL 103 Superelement

  o Error message: SOL 103 Response Simulation, SOL 200 Model Update, SOL 103 with the **Solution Process** menu set to either **Correlation** or **Laminate Dynamics**

To obtain a 64-bit binary file for NX Nastran, you should solve the input file with the ILP64 version of NX Nastran. In NX 10, this is the version that the `UGII_NX_NASTRAN` environment variable references by default.

For MSC Nastran input files that contain large ID values, NX automatically exports the associated bulk data entries using the large field format. NX does not do any other special processing when it exports MSC Nastran input files. This is because MSC Nastran does not require any specific bulk data entries or system cells to handle large integer ID values.

Control over the use of THRU formatting

In previous releases, when you exported a Nastran input file, the software always took advantage of the Nastran `THRU` format for applicable bulk data entries. In general, using the `THRU` format reduces the size of the exported input file.

Now, you can use the new **Use THRU Format** option on the **Formatting options** tab to control whether you want to use the `THRU` format. If you clear this check box, NX will not use the `THRU` format when it exports the following bulk data entries:

- `ACCEL1`
- `ASET1`
- `BLSEG`
- `BNDFIX1`
- `BNDFREE1`
- `BOLT`
- `BSET1`
- `BSURF`
- `CSET1`
- `GROUP`
- `EBDSET`
- `MATCID`
- `NSM1`
- `NSML1`
- `OMIT1`
- `PLOAD2`
- `PLOAD4`
- `QBDY3`
- `QSET1`
- `QVOL`
- `RADBC`
- `ROTORB`
- `ROTORG`
- `RVDOF1`
- `SPC1`

- `SPOINT`
- `SUPORT1`
- `TEMPP1`
- `TEMPRB`
- `USET1`

Note

The **Use THRU Format** option applies only to bulk data entries and does not affect how the `SET` case control command is exported.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file as the displayed part and the work part NX Nastran or MSC Nastran as the specified solver
Menu	**File→Export→Simulation**

Controlling the organization of an exported input file

What is it?

When you use export a Nastran input file from NX, you can now control how the data in that exported file is organized. Beginning in this release, you can specify a *pattern* file for the software to use as a guide for organizing data during the export operation. A pattern file is a file that you have previously exported from NX and might have rearranged to your preference.

For example, you can use a pattern file to establish the following:

- The order in which you want NX to export entities, such as parameters. For example, you can create a pattern file in which the Nastran `PARAM` cards always appear at the end of the bulk data section of the input file.
- How included files are used during the export. For example, if you always want to export the `GRID` bulk data entries that define nodes in a separate file, you can create a pattern file in which the nodal data is stored in a separate grid.dat file and simply referenced in the pattern file with an `INCLUDE` statement.
- The format to use for bulk data entry fields.

After you create a pattern file, you can use options in the **Round Trip Parameters** modeling object dialog box to specify the following:

- The pattern file that you want to scan for information and organizational structure.
- Whether you want to export data using the same included files as the pattern file.

When you later export a solution that references this **Round Trip Parameters** modeling object that you specify on the **General** tab in the **Solution** dialog box, NX uses the pattern file to configure the data in the exported file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part An NX Nastran or MSC Nastran solution active
Command Finder	**Modeling Objects**

Creating node and element groups from an appended file

What is it?

When you append a Nastran input file to an existing FEM or Simulation file, you can use the new **Create group from appended file** option to create a group of the nodes and elements that NX imports from the input file. This option is available when you select **Append** or **Append Merge** from the **Import Behavior** list in the **Solver Deck Append** dialog box.

During the import and append process, NX checks the ID of each imported nodes and element with the IDs of the nodes and elements in the existing FEM and Simulation file.

- If NX does not find a conflict between the imported ID and the existing ID values, NX adds the node or element to the group.

- If NX finds a conflict, NX uses the options you select in the **Merge Conflict Resolution Options** in the **Solver Deck Append** dialog box to determine whether to include the node or element in the NX group.

  - If you select **Ignore File Data**, NX does not add the node or element to the group.

  - If you select **Modify With File Data**, NX adds the node or element to the group.

  - If you select **Add File Data**, NX adds the node or element to the group and uses the specified **Label Offset** value to increment the ID value from the solver input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file as the displayed part and the work part NX Nastran or MSC Nastran as the specified solver
Menu	**File→Append**
Location in dialog box	**General Options** group in the **File Append** or **File Merge** dialog box

Import and export support improvements for Nastran

What is it?

This release includes support enhancements for Nastran:

- Bulk data entries and case control commands
- Parameters
- System cells

Bulk data and case control command support updates

The following table details the changes in bulk data and case control command support for this release.

Name	NX Nastran import/export support	MSC Nastran import/export support	Notes
ACMODL	Yes	No	Defines modeling parameters for the interface between the fluid and the structure.
ACSRCE	Yes	No	Defines source strength as a function of frequency for a simple acoustic source.
ADAPTERR case control command	Yes	No	In the NX Multiphysics environment, controls the computation and output of error estimates for adaptive meshing. See Adaptive meshing enhancements for more information.
BGPARAM	Yes	No	SLIDE is now supported
BGSET case control command	Yes	No	The STATUS option is now supported.

BOLT	Yes	No	ETYPE=3 is now supported. See Bolt Pre-Load enhancements for more information.
CRSTRN	Yes	No	
DAREA	Yes	Yes	See Darea Nodal Force and Moment load and Acoustic source load for more information.
DELAY	Yes	Yes	See Dynamic load time delay for more information.
DPHASE	Yes	Yes	See Dynamic load phase lead.
DTEMP	Yes	No	
DTEMP case control command	Yes	No	
DTEMPEX	Yes	No	
ELSTRN case control command	Yes	No	Requests elastic strain at grid points on elements.
FLSTCNT case control command	Yes	No	
GCRSTRN case control command	Yes	No	
GELSTRN case control command	Yes	No	Requests elastic strain at gauss points.
GPLSTRN case control command	Yes	No	Requests grid point strains for printing only.
GSTRAIN case control command	Yes	No	Requests strain at gauss points.
GSTRESS case control command	Yes	No	Requests stress at gauss points.

GTHSTRN case control command	Yes* (For the NX Multiphysics environment only)	No	Requests thermal strain at gauss points. Gauss point output is only available in the NX Multiphysics environment. See Additional structural outputs.
MAT10	Yes	Yes	NX now supports the GAMMA, TIDBULK, TIDRHO, TIDGE, and TIDGAMMA fields.
MATCRP	Yes	No	For more information, see Norton-Bailey Power Law creep support in NX Multiphysics.
MATOVR	Yes	No	
MATS1	Yes	Yes	Accompanies the MAT1, MATG, MATHP cards. In NX900 upgrade from FAM Type 1 to FAM type 3. In NX10 on import, set to Undetermined
MODCON case control command	Yes	No	The n to use a set to define modal contributions is not supported by NX. Additionally, the PANELMC field can reference only a single PANEL.
MPC	Yes	Yes	For more information, see Import improvements for Nastran.
NLCNTL	Yes	No	

`OPRESS` case control command	Yes	No	Supported in the NX Multiphysics environment for **Coupled** analyses only. For more information, see Additional structural outputs.
`OTEMP` case control command	Yes	No	Supported in the NX Multiphysics environment only. Supported for import since NX 9 but only documented in NX 10. For more information, see Additional structural outputs.
`PANCON` case control command	Yes	No	The `n` to use a set to define modal contributions is not supported by NX. Additionally, the `PANEL` field can reference only a single `PANEL` bulk data entry.
`PANEL`	Yes	No	You can import panels that are defined with nodes, elements or physical property tables.
`PBEAR`	Yes	No	See PBEAR physical property table enhancements for more information.

PGPLSN	Yes	No	Supported in the NX Multiphysics environment only. For performance reasons, the GRID IDs that you specify on this bulk data entry are imported into NX as **Mesh Associated Data**.
PLSTRN case control command	Yes	No	Supported in the NX Multiphysics environment only. For performance reasons, the GRID IDs that you specify on this bulk data entry are imported into NX as **Mesh Associated Data**.
RESVEC	Yes	Yes	For NX Nastran, the NORVEL field is not supported for import. For MSC Nastran, the NOADJLOD field is not supported for import.
RIGID case control command	Yes	Yes	Adds support for the AUTO and STIFF fields in the NX Multiphysics environment.
SEQDEP case control command	Yes	No	Supported for import since NX 9 but only documented in NX 10.
SET1	Yes	No	Processed in the context of a reference by the ACMODL card
SET3	Yes	No	Processed in the context of a reference by the ACMODL card
TABLED3	Yes	Yes	EXTRAP is not supported?

TABLEST	Yes	Yes	Prior to NX 10, an imported TABLEST entry could only reference a TABLES1table. Now, the TABLEST entry can also reference TABLED1, TABLED2, or TABLED3 entries. This works if the TABLEST entry from a PBEAR entry.
TEMPEX	Yes	No	Supported in the NX Multiphysics environment only.
THSTRN case control command	Yes	No	For more information, see Additional structural outputs.
TSTEP1	Yes	No	Supported in the NX Multiphysics environment only. Only the last Tendi, Ninci, and Nend in the entry are imported into NX.

Parameters

The following table details the changes in parameter support for this release.

Parameter	Description	Notes
PARAM,ROTCSV	Specifies CSV output for Rotordynamics solutions.	You can now import an NX Nastran input file that contains PARAM,ROTCSV for either a SOL 107 or a SOL 110 analysis.
PARAM,UNITSYS	Identifies the units system to use.	Support has been updated to include temperature units. NX writes out PARAM,UNITSYS only when the specified temperature unit is different from the default (°C is default for metric and °F is default for British).

System cell support updates

The following table details the changes in system cell support for this release.

System Cell	Name	Description	Notes
119			MSC Nastran system cell
281	SPLINE_METRICS	Print additional quality information for splines.	MSC Nastran system cell
414		Controls whether the lumped mass matrix contains translational components only for CBAR and CBEAM elements	MSC Nastran system cell
443	ESNLRO	Controls Nonlinear Response Optimization with the concept of Equivalent Static Loads.	MSC Nastran system cell
444		Controls whether MSC Nastran uses the pre-Nastran 2010 or the 2010 release of IFP for bulk data processing.	MSC Nastran system cell
445			MSC Nastran system cell
	IFPSTAR	Controls whether MSC Nastran uses the pre-Nastran 2010 or the 2010 release of Common Data Model for data processing.	
445	MNLQ4C	Allows or disallows corner stress calculations for material nonlinear CQUAD4 elements.	MSC Nastran system cell
449	PARSLAVE	Controls the DMP execution of slave processes in nonlinear analysis	MSC Nastran system cell

451		If set non zero, requests the method to calculate transverse shear correction for elements using MID4, or “Z0” on the PCOMP to offset a shell element. The default 0 uses a method that avoids excessive transverse shear stiffness. This is expected to have small effect on most models, unless the shell is thick and the offset is large.	MSC Nastran system cell
579	FREQVM	Determines whether the von Mises stress and strain are computed for a deterministic frequency response analysis in SOL 108 or SOL 111.	NX Nastran system cell
580		Suppresses warning messages from the DMAP compiler that arise because of undefined input or output to DMAP commands.	NX Nastran system cell
581		Turns off the error trap which occurs when an SPCD entry is defined with a bolt pre-load.	NX Nastran system cell
583		Controls the maxminum number of flutter iterations in SOL 145.	NX Nastran system cell
587		Determines whether axisymmetric input and output is on a per radian or per 2π radian basis.	NX Nastran system cell
589		Controls the R6 stiffness for CQUADR and CTRIAR elements.	NX Nastran system cell
624	IFPBUFF	Specifies the size of IFPSTAR data base I/O transfers	MSC Nastran system cell
653	NONUPIV		MSC Nastran system cell
655	GPU_MIN_FRONT		MSC Nastran system cell
656	GPU_MIN_RANK		MSC Nastran system cell

662			MSC Nastran system cell
664	SEGLOG		MSC Nastran system cell
666	CNTBKCMP		MSC Nastran system cell
676	LMFBKCMP		MSC Nastran system cell
677	FFOBKCMP		MSC Nastran system cell

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file as the displayed part and the work part NX Nastran or MSC Nastran as the specified solver
Menu	**File→Import→Simulation** **File→Export→Simulation**

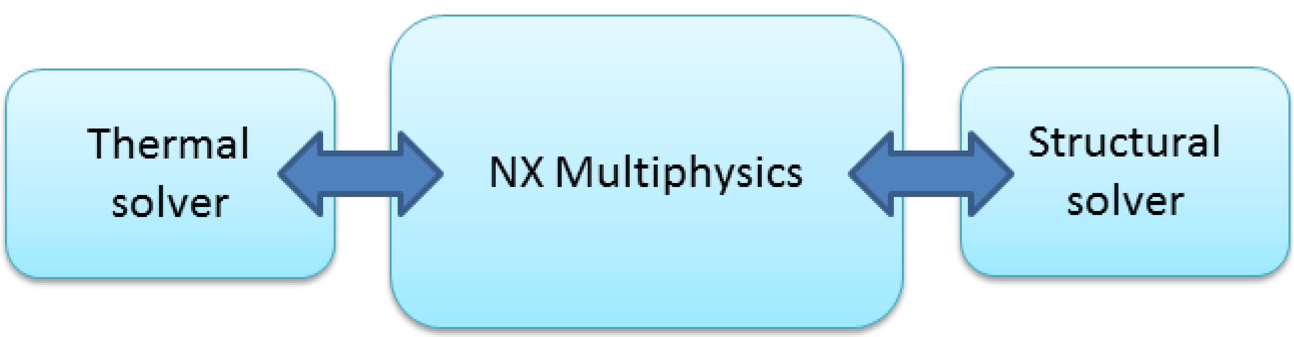
NX Multiphysics enhancements

Coupled thermal-structural analysis

What is it?

You can now perform a coupled thermal-structural analysis in the NX Multiphysics environment. This is in addition to the **Structural**, **Thermal**, and **Mapping** analysis types that were available in the previous release of the NX Multiphysics environment.

The **Coupled** analysis type links a structural solution from the NX Nastran SOL 401 Multi-Step Nonlinear solver and a thermal solution from the NX Thermal solver. The results from the thermal solution, for example, temperatures, affect the structural solution, and the results from the structural solution, for example, displacements, affect the thermal solution.



Data is passed between the solvers through the NX Multiphysics application

The thermal solver can pass the following data to the structural solver:

- Temperatures on nodes and elements

- Thermal boundary condition pressures. NX maps pressures on 1D or 2D thermal elements to pressures on structural faces and edges.

The structural solver can pass the following data to the thermal solver:

- Nodal displacements

- Contact pressures and contact separation (gap) distance on nodes. The contact pressures can be used to determine conductive properties at the contact surfaces.

You use the same mesh for both the structural and thermal solution.

As you do with the other NX Multiphysics analysis types, you can use a condition sequence to create the solution and solution steps.

Solution steps

The sequence of solution steps is based on the **End Time** that you define for each solution step. By default, a solution step is sequentially dependent on the results of the previous solution step, and its start time is the **End Time** of the previous solution step. When you create a new solution step in a coupled analysis solution, you can define the step as one of these types.

Thermal Nonlinear Statics

Coupled thermal-structural step in which the structural solver and the thermal solver exchange data through the NX Multiphysics application.

In each solution step, you can specify that the thermal solution type is transient or steady state, and you can specify nonlinear control parameters for the structural solver.

The structural solver runs a static solution in which loads can be constant or time assigned.

This type of solution step can include geometry and plasticity or creep material nonlinearity.

Normal Modes

Specialized structural step that runs a modal analysis. This step can include stress stiffening, follower stiffness, and the spin softening from the previous thermal nonlinear static solution step.

Preload

Specialized structural step for a **Bolt Pre-Load**, including contact and glue, or **Temperature** load. Preload steps must have an **End Time** of **0**. Any solution step that precedes a **Preload** step must also have an **End Time** of **0**. You can have only one **Preload** step in a solution.

Time step control

Within a **Thermal Nonlinear Statics** or **Preload** solution step, the thermal solver and the structural solver perform their respective solves at each time step. The thermal solver is always the starting solver. You can control the time steps taken by each solver separately. You control the time steps in a solution step by specifying the **End Time** and **Number of Increments** for that step.

The time steps for a solution step correspond to the `TSTEP1` NX Nastran bulk data entry.

Alternatively, with **Thermal Nonlinear Statics** steps, you can specify a constant **Step Size** in terms of time. By default, the thermal solver uses **Automatic** adaptive time stepping.

Coupling times

The structural solver and the thermal solver exchange data at coupling times. You can control the time step increment at which NX Multiphysics exchanges data between the solvers, up to the solution step **End Time**. The **End Time** is always a coupling time. By default, the coupling times are at each of the thermal solver time steps. To change the coupling time basis, you can use the **Coupling Time Option** in the **Coupled Solution Parameters** for the solution. Coupling times are added automatically as common integration time points for both solvers.

At each coupling time, temperatures on nodes and elements (from the thermal solution) are passed to the structural solver. You can specify that additional types of data are passed at each coupling time using the options in the **Coupled Solution Parameters** for the solution.

Sequential or iterative coupling

In the **Coupled Solution Parameters** for the solution, you can specify the coupling **Mode**:

Sequential — The structural solver and the thermal solver pass information to each other once per coupling time and do not attempt to converge together. NX solves the first solution until convergence is obtained, and then solves the other solution using the values from the first solution.

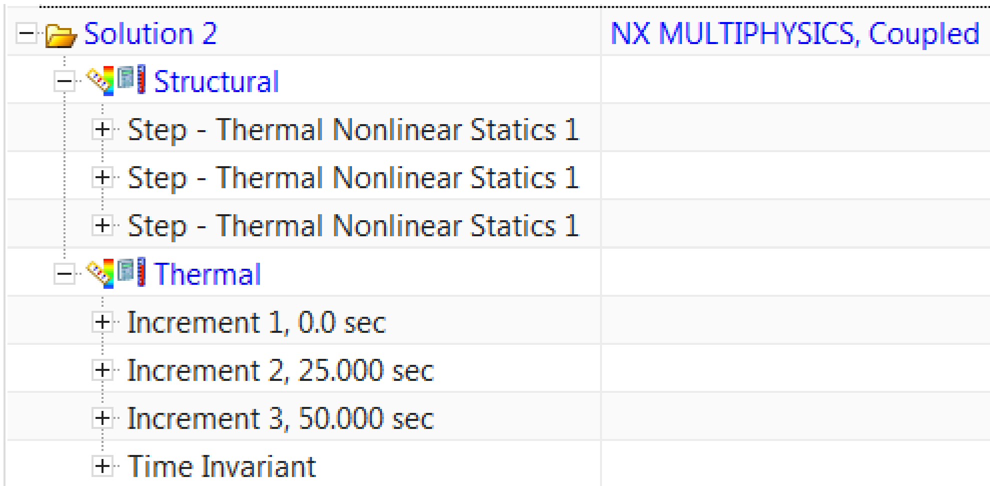
Iterative — The structural solver and the thermal solver pass information to each other for several iterations, until the variables that are being passed converge. This method requires you to specify the **Iteration Limit**, **Convergence Criterion**, **Convergence Norm**, and **Relaxation** values. Coupled convergence occurs when both temperatures and displacements have converged.

Output of solution data

During the course of a coupled analysis, the structural solver and the thermal solver perform many solves. You may want to output data for post processing at each of these solution points or only at certain points, such as at the **End Time** of the solution step. You can control this output in the **Solution Step** dialog box on the **Time Step Definition** page, under the **Output Control** group.

Post-processing structural and thermal results

In NX Multiphysics solutions, the structural and thermal results are listed together under the same solution in the **Post Processing Navigator**.



Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Coupled	Multi-Step Nonlinear

Where do I find it?

Application	Advanced Simulation
Location in dialog box	**Solution** dialog box→**Analysis Type**=**Coupled**

Adding time points to a structural solution to match a reference solution

What is it?

The **Augmented Time Step List** modeling object is added to let you import additional time points into an NX Multiphysics structural solution.

With NX Multiphysics, you can create structural solutions in which the temperatures are mapped from a thermal solution. If the thermal solution used automatic time stepping, the time stepping is uneven. If you want to solve the structural solution at the same time points as those specified in the thermal solution, you must add those time points to the structural solution. To do this, you can import the uneven list of time points from the thermal results into the structural solution. NX Nastran then solves the structural solution at the same time points as the thermal solution, along with any time points you specify in the structural solution.

You can import these additional time points from an NX Nastran, NX Thermal, Abaqus, or ANSYS results file that contains transient results. If you want to subdivide the additional time points, you can specify a number of increments. You can also specify at what point the solver should output data, for example, at the solution step end time or at each time step increment.

You can also import time points from a CSV file.

You associate the **Augmented Time Step List** with the solution in the **Solution** dialog box on the **Solution Control** page.

When you solve, NX writes the additional time points to the `TSTEP1` bulk data entry in the NX Nastran input file.

Example

The following NX Nastran input file shows the time points in the structural solution, before adding the augmented time step list.

```
$* SOLUTION CARDS
$*
$*  NX Modeling Object: Nonlinear Control Parameters1
NLCNTL       100 MAXITER      50  MSGLVL       1
BGADD        100     101     102
$*  NX Load and Constraint: Edge-to-Edge Gluing(1)
BGSET        101       9     101.000000
$*  NX Load and Constraint: Edge-to-Edge Gluing(2)
BGSET        102       1      31.000000
TSTEP1       101  0.0000       1      YES
TSTEP1       102 60.0000       1      YES
TSTEP1       103780.0000       1      YES
TSTEP1       104833.4000       1      YES
```

This example shows additional time points imported from an NX Thermal .bun file with the augmented time step list.

```
$* SOLUTION CARDS
$*
$*  NX Modeling Object: Nonlinear Control Parameters1
NLCNTL       100 MAXITER      50  MSGLVL       1
BGADD        100     101     102
$*  NX Load and Constraint: Edge-to-Edge Gluing(1)
BGSET        101       9     101.000000
$*  NX Load and Constraint: Edge-to-Edge Gluing(2)
BGSET        102       1      31.000000
TSTEP1       101  0.0000       1      YES
TSTEP1       102 16.2500       1      YES
+                23.6364       1      YES
+                36.0000       1      YES
+                44.0000       1      YES
+                50.0000       1      YES
+                58.0000       1      YES
+                60.0000       1      YES
TSTEP1       103 74.1176       1      YES
+                88.2353       1      YES
+               102.3529       1      YES
+               120.6677       1      YES
+               586.0787       1      YES
+               683.0394       1      YES
+               780.0000       1      YES
TSTEP1       104780.7105       1      YES
+               781.4210       1      YES
+               781.4280       1      YES
+               781.5314       1      YES
+               821.7022       1      YES
+               827.5511       1      YES
+               833.4000       1      YES
```

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Structural	Multi-Step Nonlinear

Where do I find it?

Application	Advanced Simulation
Location in dialog box	**Solution** dialog box→**Solution Control** page

Step Association Manager

What is it?

The **Step Association Manager** command is now available in the NX Multiphysics environment. Use this command to:

- Manage which boundary conditions are associated with a given solution step.
- Change multiple boundary condition assignments across solution steps.
- Add a scale factor to a load in a particular solution step.
- Export information about the boundary condition assignments to an HTML page or a spreadsheet, make changes in the spreadsheet, and import the changes back into NX.

For small models, and during the initial creation of the model, the standard methods of assigning individual boundary conditions to the active solution or step may be adequate. These methods, however, may become unwieldy when you want to edit or refine a model that contains a large number of defined boundary conditions, boundary conditions that are organized into multiple folders, simulations that contain multiple solutions, and solutions that contain multiple solution steps or subcases. Use the **Step Association Manager** to manage and document the distribution of multiple boundary conditions across steps within a solution.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Structural	Multi-Step Nonlinear
	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Command Finder	**Step Association Manager**

New boundary conditions supported in NX Multiphysics

What is it?

NX Multiphysics has added support for the following loads and boundary conditions:

- **Moment**
- **Enforced Motion Load**
- **Enforced Displacement**
- **Surface-to-Surface Contact**

- **Edge-to-Edge Contact**
- **Edge-to-Edge Gluing**
- **Material Override**
- **Override Set — Thermal Properties**
- **Solid Motion Effects**

Some loads and constraints that were previously available only in structural or only in thermal solutions are now available in both. You can create **Surface-to-Surface Gluing** in thermal solutions. **Thermal Convecting Zone**, **Thermal Stream**, and **Thermal Void** loads that you create in thermal and coupled solutions, can be added to structural solutions to define pressure only. You can use all of the loads and constraints for structural and thermal solutions in the new **Coupled** type of analysis.

Initial Temperatures are no longer available as simulation object types. You now define them with temperature loads and a temperature set. For more information on defining initial temperatures, see Updated processes for defining temperatures in a Nastran analysis.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver

Controlling plasticity and creep effects

What is it?

You can control plasticity and creep effects at both the solution and the step level.

Solution level

In the **Solution** dialog box, the **Material Nonlinearity** option must be turned on for any plasticity or creep effects to exist in the solution. You can control creep and plasticity using a **Nonlinear Control Parameters** modeling object.

You can also disable the creep and plasticity effects for individual elements or for physical property tables by including a **Material Override** simulation object in the solution.

Step level

In the **Solution Step** dialog box, you can separately control creep and plasticity using a **Nonlinear Control Parameters** modeling object.

Corresponding Nastran syntax

The **Material Nonlinearity** option for the solution corresponds to the NX Nastran `MATNL` parameter.

The **Nonlinear Control Parameters** for creep and plasticity correspond to the NX Nastran `CREEP` and `PLASTIC` parameters on the `NLCNTL` bulk entry.

The **Material Override** simulation object corresponds to the NX Nastran `MATOVR` bulk entry.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Structural	Multi-Step Nonlinear
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver Structural or Coupled as the specified analysis type
Command finder	**Material Override**
Location in dialog box	**Solution** dialog box → **Solution Control** page → **Material Nonlinearity** check box **Solution** dialog box → **Solution Control** page → **Nonlinear Control Parameters** list → **Plasticity Effects** or **Creep Effects** **Solution Step** dialog box → **Step Control** page → **Nonlinear Control Parameters** list → **Plasticity Effects** or **Creep Effects**

Norton-Bailey Power Law creep support in NX Multiphysics

What is it?

You can now use the Norton-Bailey Power Law creep model in Multi-step Nonlinear solutions when NX Multiphysics is the specified solver.

You select this model in the **Materials** dialog box, on the **Creep** page, using the **Temperature-Dependent Time Hardening (Norton-Bailey) Power Law (Type 301)** option. This option corresponds to the new `MATCRP` bulk data entry, which is an alternate method for selecting the Norton-Bailey creep model. It lets you define the coefficients of the power law equation as either constant or temperature dependent.

$$\varepsilon_e^c = A\sigma_e^B t^D$$

In the **Materials** dialog box, you define the A, B, and D coefficients of this creep model in the **Constant Multiplier (a)**, **SIGMA Exponent (b)**, and **Hardening Exponent (d)** boxes.

- To define creep at a single temperature, enter real values for the coefficients.
- To define creep as temperature dependent, define a table field for the coefficients. The table corresponds to the TABLEM1 bulk data entry.

Checking system units

A **Check Unit System** option is added to the new Type 301 power law as well as to the existing Time Hardening (Norton-Bailey) Power Law Type 300. This option checks for inconsistent units in the solution and the creep coefficients. NX checks when the solver input file is exported and alerts you if the solution units are not consistent with the creep coefficient units. NX cannot convert the creep coefficient units when it exports the solver input file.

Controlling creep effects

For NX Multiphysics, you can now turn off creep effects at the solution level and subcase level. For more information, see Controlling plasticity and creep effects.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver and an active Multi-step Nonlinear solution or NX Nastran as the specified solver and an active SOL 601,106 Advanced Nonlinear Statics solution
Command Finder	**Manage Materials**
Location in dialog box	**Create Material** →**Creep** page

Support for plastic strain material nonlinearity in NX Multiphysics

What is it?

In the NX Multiphysics environment, you can now define a stress-dependent material with plastic strain properties. In the **Materials** dialog boxes, on the **Mechanical** page, a new **Type of Nonlinearity** option **PLSTRN** has been added.

This option corresponds to the `MATS1` bulk data entry with `TYPE=PLSTRN`. For more information, see *MATS1* in the *NX Nastran Quick Reference Guide*.

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver An active Multistep Nonlinear solution
Command Finder	**Manage Materials**
Location in dialog box	**Create Material** →**Mechanical** page→**Stress-Strain Related Properties** group→**Type of Nonlinearity**=**PLSTRN**

Support for 2D seed mesh creation in NX Multiphysics

What is it?

In the NX Multiphysics environment, when you create a mesh for a structural analysis, you can now create 2D seed meshes to control the placement of nodes on subsequent 3D meshes. In the **2D Mesh** or the **2D Mapped Mesh** dialog box, you can select either the **Linear Quadrilateral Seed** or **Linear Triangle Seed** options from the **Type** list to create the seed mesh. These seed meshes have the following characteristics:

- They can only be used to create solid meshes.
- They do not have any associated physical properties or mesh associated data.
- They are not written out to the solver input file when you export or solve your model.

In previous releases, you could create only 3D meshes for models in which NX Multiphysics was the specified solver.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and work part NX Multiphysics as the specified solver **Structural** as the specified analysis type
Command Finder	**2D Mesh** or **2D Mapped Mesh**
Menu	**Insert→Mesh→2D Mesh**

Support for the generalized plane strain formulation

What is it?

NX Multiphysics now supports the new generalized plane strain element formulation, which was introduced in the NX Nastran 10 release. You can use this new formulation with the following plane strain element types:

- **Plane Strain Linear Triangle** (`CPLSTN3`)
- **Plane Strain Linear Quadrilateral** (`CPLSTN4`)
- **Plane Strain Parabolic Triangle** (`CPLSTN6`)
- **Plane Strain Parabolic Quadrilateral** (`CPLSTN8`)

In NX, you use the **Generalized Plane Strain** physical property table to specify this new element formulation and associate it with the elements. The options in this dialog box correspond to the NX Nastran `PGPLSN` bulk data entry. For example, you can specify:

- The isotopic (`MAT1`) or orthotropic (`MAT3`) material to use.

- The element’s thickness in its undeformed state.
- Optional additive normal stiffness and rotational stiffness values.

Defining a controlling node to add additional degrees-of-freedom

With the `PGPLSN` entry in NX Nastran, you can define a control grid point (node) that adds three additional degrees-of-freedom to the element:

- One translational degree-of-freedom along the element’s thickness direction.
- Two rotational degrees-of-freedom around the element’s two in-plane axes.

The control grid point is the location where NX Nastran applies any out-of-plane loads or enforced displacements to the associated elements.

In NX, you can use the **Generalized Plane Strain Control Node** option in the **Mesh Associated Data** dialog box to select the appropriate node to use as the control grid point.

Generalized plane strain formulation is for specialized applications

The generalized plane strain formulation is applicable only to small strain, small deflection structural analyses. These structural analyses include linear static, creep, and plasticity analyses, and combination creep and plasticity analysis. Analysis with the generalized plane strain formulation is highly specialized and is typically used to evaluate the behavior of gas turbine compressor and turbine blades. For such an analysis, you mesh the cross section of the blade with the appropriate plane strain elements and then create a **Generalized Plane Strain** physical property table to define the properties for those elements.

Where do I find it?

Generalized Plane Strain physical property table

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part NX Multiphysics as the specified solver **2D Solid Option** set to **ZX Plane, Z Axis** or **XY Plane, X Axis**
Command Finder	**Physical Properties**

Mesh Associated Data

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part NX Multiphysics as the specified solver **2D Solid Option** set to **ZX Plane, Z Axis** or **XY Plane, X Axis**
Command Finder	**Mesh Associated Data**

Checking the completeness of NX Multiphysics solutions

What is it?

You can now use the **Model Setup** command to check the completeness of structural and coupled thermal-structural solutions in the NX Multiphysics environment before you solve. In the NX Multiphysics environment, the **Model Setup** commands checks to ensure that:

- A material is specified.
- The material orientation is defined.
- A physical property table for the elements is specified.
- The model has loads or constraints applied.

For coupled thermal and structural solutions, **Model Setup** performs some checks specific to thermal analyses, such as the following:

- Whether the appropriate element data is specified.
- Whether the appropriate modeling objects are specified when required.
- Whether all geometry is meshed.

Additionally, if the model contains laminate composites, the **Model Setup** check issues warnings or error messages related to the laminates, if necessary.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file as the displayed part and the work part NX Multiphysics as the specified solver
Command Finder	**Model Setup**

Additional structural outputs

What is it?

You can now request contact results and additional strain results for an NX Multiphysics structural and coupled analysis.

For contact results, you may choose to include the contact pressure, contact force, contact status, and the minimum value of separation distance between source and target elements.

The following stress and strain results are now available:

- **Creep Strain**
- **Plastic Strain**
- **Elastic Strain**

- **Thermal Strain**
- **Gauss Point Creep Strain**
- **Gauss Point Plastic Strain**
- **Gauss Point Elastic Strain**
- **Gauss Point Strain**
- **Gauss Point Thermal Strain**
- **Gauss Point Stress**

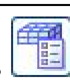
To control whether these options are selected by default, use the **Results Options** tab for NX Multiphysics in the **Customer Defaults** dialog box.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Structural	Multi-Step Nonlinear
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Structural Output Requests

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver
Command Finder	**Modeling Objects** 
Location in dialog box	**Type** list →**Structural Output Requests**

Results Options customer defaults

Menu	**File** → **Utilities** →**Customer Defaults**
Location in dialog box	**Simulation** →**NX MULTIPHYSICS** → **Results Options**

Defining thermal coupling with contact and gluing

What is it?

You can now define heat transfer through contact and gluing simulation objects for **Coupled** and **Thermal** types of NX Multiphysics analysis. You do this using the **Activate Thermal Coupling** option in the **Edge-to-Edge Contact**, **Edge-to-Edge Gluing**, **Surface-to-Surface Contact**, and **Surface-to-Surface Gluing** dialog boxes.

By specifying the thermal coupling type, you determine what thermal properties must be specified:

- Select **Perfect Contact** to transfer heat with no thermal resistance.

- Select **Heat Transfer Coefficient** to define the heat transfer coefficient for the contact pair.
- Select **Edge Contact** to define the conductance per length of the edge.
- Select **Thermo-Mechanical Contact** to use one heat transfer coefficient when there is a gap between the contact pair, and a separate heat transfer coefficient when the contact pair has no gap.

The following table shows which thermal coupling types are available for each simulation object.

Simulation object type	Thermal coupling types available
Edge-to-Edge Contact	**Perfect Contact** **Heat Transfer Coefficient** **Edge Contact** **Thermo-Mechanical Contact**
Edge-to-Edge Gluing	**Perfect Contact** **Heat Transfer Coefficient** **Edge Contact**
Surface-to-Surface Contact	**Perfect Contact** **Heat Transfer Coefficient** **Thermo-Mechanical Contact**
Surface-to-Surface Gluing	**Perfect Contact** **Heat Transfer Coefficient**

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Structural	Multi-Step Nonlinear
	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver

Command Finder	**Edge-to-Edge Contact** **Edge-to-Edge Gluing** **Surface-to-Surface Contact** **Surface-to-Surface Gluing**

Preload solution step

What is it?

You can now create a **Preload** type of solution step for structural and coupled NX Multiphysics solutions.

Bolt Pre-Load and a **Temperature Load** type of temperature set are the only loads that you can include in the preload solution step. You can add an optional constraints container to the solution step, and add any structural constraints to that.

The **Preload** solution step is the only step in an NX Multiphysics solution where you can add a **Bolt Pre-Load** Load. You cannot add it to other types of solution steps or directly to the solution load container.

In the **Time Step Definition** of the preload step, the **End Time** is always 0. You can use multiple increments if the solution does not converge correctly with the **Number of Increments** set to 1.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Structural	Multi-Step Nonlinear
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

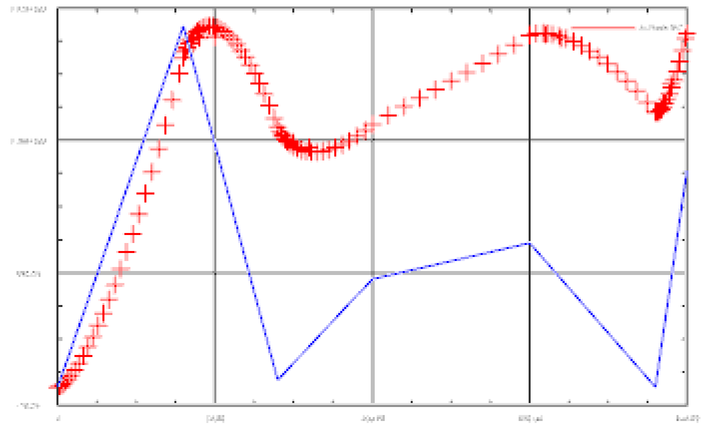
Where do I find it?

Application	Advanced Simulation
Prerequisites	NX Multiphysics as the specified solver
Location in dialog box	**Solution Step** dialog box → **Step** list → **Step — Preload**

Adaptive time stepping enhancement

What is it?

The new adaptive time stepping scheme handles the sharp changes in temperature at boundary conditions better than the previous scheme. When there are no abrupt changes in boundary conditions, it accelerates the simulation without losing accuracy. The time step size calculation is now based on the estimated error between the analytical solution and the simulation. The error is the difference between a fit quadratic function and a linear variation through three consecutively computed temperature values for two consecutive time steps.



When there are abrupt temperature changes in the model, as seen in the example, the adaptive time stepping scheme adapts by performing smaller time steps around the times when the abrupt changes occur. The blue curve represents the time-varying heat load that is applied to the boundary condition and each red + represents the temperature value at one point of the boundary condition for the transient run. The +'s that are close to each other indicate that the time steps are smaller at those times to better capture the changes in the heat load.

You specify the temperature error tolerance, the minimum time step size, and the maximum time step size.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Command Finder	**Step-Subcase**
Simulation Navigator	Right-click a thermal step node→**Edit**
Location in dialog box	**Time Step Definition** node→**Time Step Control** group→**Time Step Option** list→**Automatic**

Solution Monitor enhancements

What is it?

- The **Solution Monitor** is now used by the NX Multiphysics and NX Nastran solvers.
- The **Solution Monitor** now displays text output for each solver on a separate tab when you run a coupled NX Multiphysics analysis.

- o The **NX Multiphysics** tab displays the text output for the coupled solver.
- o The **NX Thermal** tab displays the text output for the thermal solver.
- o The **NX Nastran** tab displays the text output for the NX Nastran solver.

- You can copy and paste the displayed text output. In previous versions, the displayed text output was not selectable.
- The user interface of the **Solution Monitor** is refreshed and the command names are more intuitive. The commands now have tooltips describing their functionality.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	NX Nastran, NX Multiphysics, NX Thermal and Flow, NX Space Systems Thermal, or NX Electronic Systems Cooling as the selected solver environment
Command Finder	**Solve**
Simulation Navigator	Right-click a solution node→**Solve**

Multi-layer shell support

What is it?

You can now use multi-layer shell elements to model thin geometries, such as panels and layers, in the NX Multiphysics solver environment.

In the **Thin Shell** physical property table, you can now specify a thin shell to be treated as a multi-layer shell by selecting the **Divide Thickness into Uniform Layers** option. NX divides the shell thickness that you specify equally into a specified number of layers.

When you define multi-layer shells, you can now select one or more layers to which you apply the boundary condition in the **Temperature** and **Thermal Loads** dialog boxes.

Why should I use it?

Using multi-layer shell elements, you can obtain a detailed picture of the conduction through the geometry without using thin solid (3D) elements. Multi-layer shell elements present a smaller conductance matrix than solid elements.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal

Where do I find it?

Thin Shell physical property

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver. A FEM file as the displayed part and work part.
Command Finder	**Physical Properties**
Menu	**Insert→Physical Properties**
Location in dialog box	**Type** list→**Thin Shell**→**Create**→**Thickness** group→ **Divide Thickness into Uniform Layers**

Load and constraint

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver.
Command Finder	**Thermal Loads** or **Temperature**
Simulation Navigator	Right-click the **Loads** node→**New Load**→**Thermal Loads** Right-click the **Constraint Set** node→**New Constraint**→**Temperature**
Location in dialog box	**Multi-Layer Shells** group

Enhancements to thin shell

What is it?

The following enhancements are now available for thin shell elements in the NX Multiphysics thermal environment:

- **Material Orientation** is now added to both element associated data and mesh associated data.
- The **Thin Shell** physical property now supports orthotropic materials. In the previous release, you could choose only isotropic materials.
- You can now divide a specified thickness into equal layers in order to model uniform multi-layer shell elements in the **Thin Shell** dialog box.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver. A FEM file as the displayed part and work part.

Element and mesh associated data

Command Finder	**Element Modify Associated Data** or **Mesh Associated Data**
Location in dialog box	**Material Orientation** group

Thin Shell physical property

Command Finder	**Physical Properties**
Location in dialog box	**Type** list→**Thin Shell**→**Create**

Plane strain element support

What is it?

The NX Multiphysics thermal and coupled solutions now support the following 2D element types in both the XY and the ZX planes:

- **Plane Strain Linear Quadrilateral**
- **Plane Strain Parabolic Quadrilateral**
- **Plane Strain Linear Triangle**
- **Plane Strain Parabolic Triangle**

Although there is no element associated data related to the plain strain elements, the material orientation angle is supported as mesh associated data.

The plain strain elements are either tied to a **Plane Property** or to a **Generalized Plane Strain Property** physical property defined at the mesh collector level. In the physical property, you can specify either isotropic or orthotropic materials.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver. A FEM file as the displayed part and work part.

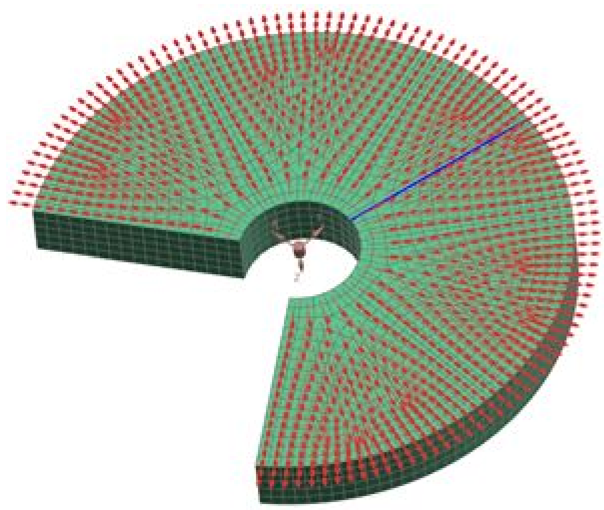
Command Finder	**2D Mesh** 
Location in dialog box	**Element Properties** group→**Type** list

Defining a thermal stream with a radial coordinate system

What is it?

You can define convection caused by radial thermal streams over surfaces. The stream flows radially over the surface starting from the origin of the specified cylindrical coordinate system.

The thermal solver creates a 1D duct with mass flow elements on the selected regions and, using convection thermal coupling, connects the 1D elements to the nearest thermal solid elements that lie in the same radial band. The 1D duct is shown in blue in the following figure.



You can set the radial **Thermal Stream** load type based on whether you want the thermal stream to convect to a single surface or to two surfaces.

In addition to specifying the region, the fluid material, the stream conditions, and the heat transfer, you also specify the cylindrical coordinate system. In the structural multiphysics solution, you only specify the pressure.

Why should I use it?

The load provides an automated way to include the effects of convection and advection in a thermal model. It improves your efficiency by regrouping many thermal simulation boundary conditions in a single command.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow

NX Multiphysics	Structural	Multi-Step Nonlinear
	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	The specified solver must be NX Multiphysics or NX Thermal and Flow.
Command Finder	**Thermal Stream**
Simulation Navigator	Right-click the **Load** container node→**New Load**→**Thermal Stream**
Location in dialog box	**Type** list→**One-Sided Stream on Faces (Cylindrical Components)** or **Two-Sided Stream on Faces (Cylindrical Components)**

Solid Motion Effects simulation object enhancement

What is it?

Use the **Spinning - All Elements** type of **Solid Motion Effects** simulation object to model the thermal effects of a spinning model. All elements in the model spin together.

You select the spinning axis, and specify the number of calculations per spin, *N*. The thermal solver calculates the view factors and thermal couplings that result from NX spinning the model *N* times, each time rotating the spinning model by 360/*N* degrees. The thermal solver then averages the view factors and thermal couplings of these *N* positions.

The view factors include the following:

- Solar, earth, and albedo view factors from orbital heating.
- Heat flux view factors from radiative heating.
- Black body and ray-traced view factors from the deterministic and hemicube methods.
- Ray-traced view factors, radiative couplings, and heat loads from the Monte Carlo method.

  Note

  Best accuracy is obtained when *N* is greater than the number of elements around the circumference of the spinning part. This is especially true for thermal coupling calculations.

The **Spinning** type of **Solid Motion Effects** simulation object from the previous releases is now called **Spinning - Selected Elements**.

Why should I use it?

The **Spinning - All Elements** type of **Solid Motion Effects** simulation object can be used, for example, to model a spinning spacecraft that spins rapidly around a specified axis.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solid Motion Effects**
Simulation Navigator	Right-click the **Simulation Object** container node→**New Simulation Object**→**Solid Motion Effects**
Location in dialog box	**Type** list→**Spinning - All Elements** or **Spinning - Selected Elements**

Varying capacitance for Void Non Geometric Element modeling object

What is it?

You can now define a time-varying or temperature-varying capacitance when you create a **Void Non-Geometric Element** modeling object. To define the capacitance, you can specify the capacitance value or the volume value. Both quantities can vary with time or temperature.

In previous releases, you could specify only a constant capacitance value.

Why should I use it?

You can use time-varying capacitance, for example, to model the capacitance of fuel in aircraft wings. The fuel mass and thus the capacitance varies with time over a flight profile.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation

Prerequisite	NX Multiphysics or NX Thermal and Flow as the specified solver.
Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Void Non-Geometric Element**→**Create**→**Calculation Method** group

Multiple Fluid material

What is it?

The new **Multiple Fluid** material type lets you define a primary fluid material and up to five additional fluid materials.

The thermal solver uses the specified primary fluid material properties during computation. The additional fluid material number can be passed as an argument to some CAE functions, such as fluid plug-ins, and also to user-defined functions. See User-defined functions for expressions for more information on plug-in functions.

You define the **Multiple Fluid** material type in the following loads and modeling object:

- **Thermal Convecting Zone** load
- **Thermal Stream** load
- **Void Non-Geometric Element** modeling object

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics or NX Thermal and Flow as the specified solver.

Defining the multiple fluid material

Command Finder	**Manage Materials**
Location in dialog box	**Type** list→**Multiple Fluid**→ **Create**

Specifying the multiple fluid material in loads

Command Finder	**Thermal Convecting Zone** or **Thermal Stream**
Simulation Navigator	Right-click the **Load** container node→**New Load**→**Thermal Convecting Zone** or **Thermal Stream**
Location in dialog box	**Fluid Materials** list

Specifying the multiple fluid material in the modeling object

Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Void Non-Geometric Element**→**Create**→**Environment** group→**Fluid Materials** list

Local temperature error estimate result sets

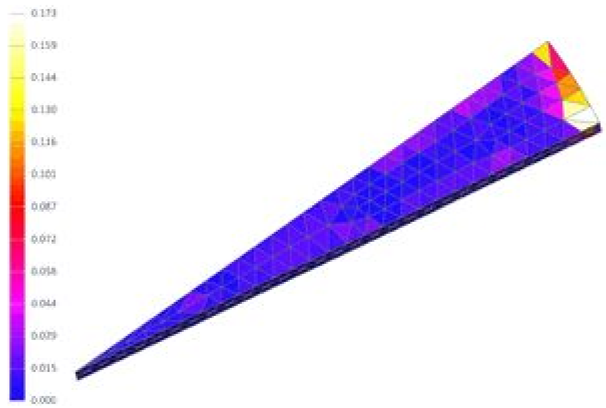
What is it?

You can now recover local temperature error estimate results. The thermal solver generates temperature error estimate results which estimate the local spatial discretization errors due to the mesh used for the simulation of conduction.

After you solve the model, the following result types appear under the solution node in the **Post Processing Navigator**:

- **Temperature Error Estimates - Nodal**
- **Temperature Error Estimates - Elemental**
- **Time of Maximum Error Estimates - Nodal**
- **Time of Maximum Error Estimates - Elemental**
- **Maximum Temp Error Estimates - Nodal**
- **Maximum Temp Error Estimates - Elemental**

The last four result types are written only for transient runs. They are displayed under the **Time Invariant** node in the **Post Processing Navigator**.



The error estimate is measured in units of temperature difference.

The elemental error estimate, ε_T, is the maximum temperature difference of all boundary elements computed as follows:

$$\varepsilon_T = \max |T_{BE} - f_{BE}(T_1, T_2, ..., T_N)|$$

where:

- T_{BE} are the calculated boundary element temperatures
- T_i are the N smoothed nodal temperatures
- N is the number of nodes
- The f_{BE} function uses element shape functions to interpolate the nodal temperatures to the boundary element

Why should I use it?

The adaptive meshing solution process uses the maximum local temperature error estimates to refine the mesh for conduction thermal analysis.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver.
Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**

Location in dialog box	**Type** list→**Thermal Output Requests**→**Create**→**Thermal** tab→ **Local Temperature Error Estimate**

Envelope results for thermal post processing

What is it?

Time invariant results from transient thermal solutions are now located under the **Time Invariant** node in the **Post Processing Navigator**.

- iron
  - Solution 1
    - Thermal
      - Increment 1, 0.0 sec
      - ...
      - Increment 61, 600.000 sec
      - Time Invariant
        - Max Temperature - Elemental
        - Min Temperature - Elemental
        - Time at Max Temperature - Elemental
        - Time at Min Temperature - Elemental

The time invariant thermal results include the following:

- Maximum and minimum elemental temperature over the complete transient run and the times at which they occur.
- Maximum thermal error estimates at each element over the complete transient run and the times at which they occur.
- Absorbed, incident, reflected, or transmitted radiative fluxes at each element when these values are constant over the complete transient run.

In previous releases, these results were displayed in the first or last time step.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal Advanced Axisymmetric Thermal
	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	Loaded results file that contains transient thermal results.

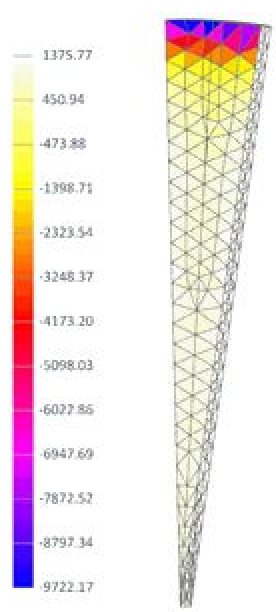
Convective heat flux results

What is it?

You can now recover convective heat flux results on 2D shell elements. You request nodal, elemental, or nodal and elemental types of results from the **Convective Heat Flux Results Location** list.

After you solve the model, depending on your request, either or both of the following result types appear under the solution node in the **Post Processing Navigator**:

- **Convective Heat Flux - Nodal**
- **Convective Heat Flux - Elemental**



Why should I use it?

Convective heat flux results help you analyze the results of your thermal simulation.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver.
Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Thermal Output Requests**→**Create**→**Thermal** tab→ **Convective Heat Fluxes**

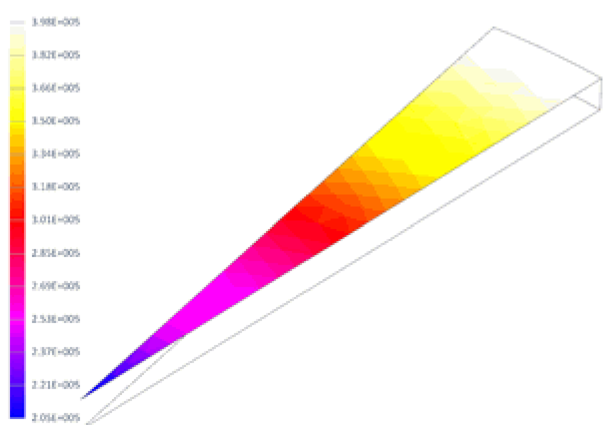
Swirl velocity results

What is it?

You can now recover swirl velocity results on 2D shell elements or on 1D axisymmetric elements. You request nodal, elemental, or nodal and elemental types of results from the **Swirl Velocity Results Location** list.

After you solve the model, depending on your request, either or both of the following result types appear under the solution node in the **Post Processing Navigator**:

- **Swirl Velocity on Walls - Nodal**
- **Swirl Velocity on Walls - Elemental**



The thermal solver computes swirl velocities when you define **Thermal Stream**, **Thermal Convective Zone**, or **Thermal Void** loads, or **Duct Flow Boundary Conditions** simulation objects of type **One-Sided Total Temperature Effects** or **Two-Sided Total Temperature Effects**.

Why should I use it?

These result sets help you analyze the behavior of your thermal model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver.
Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Thermal Output Requests**→**Create**→**1D Flow** tab→ **Swirl Velocities**

Selective thermal results

What is it?

You can request specific thermal results on selected geometry, nodes, elements, or groups. In the same thermal output request, you can specify different selections for results requests that are on the **Thermal** tab, the **Radiation** tab, or the **1D Flow** tab.

In the previous release, you could request thermal results only on all elements, all nodes, or on all elements and nodes in the model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver.

Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Thermal Output Requests**→**Create**→**Thermal** tab or **Radiation** tab or **1D Flow** tab→**Entity Selection** group→**Entity** list→**Selected**

Results location options

What is it?

The thermal solver can output the results at elements, at nodes, or at elements and nodes for additional results sets. In the previous release, the thermal solver output only elemental results for these results sets.

When you request one of the following results sets, you can now specify the location of the results.

On the **Thermal** tab:

- Total temperature
- Free and forced convection coefficient

On the **1D Flow** tab:

- Velocity
- Swirl velocity
- Pressure
- Density
- Mass flow
- Reynolds number

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics as the specified solver.
Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**

Location in dialog box	**Type** list→**Thermal Output Requests**→**Create**→**Thermal** tab or **1D Flow** tab

Heat transfer coefficient magnitude for edge interface resistance

What is it?

You can now specify the heat transfer coefficient for an **Interface Resistance** simulation object of type **Edge Interface**.

The resistance is calculated as:

1/(heat transfer coefficient x area)

At an edge interface, the area is calculated as follows:

area = (the length of the edge) x (the minimum shell thickness)

Why should I use it?

This improvement lets you define the resistance magnitude at an edge interface by specifying the heat transfer coefficient, which is a value that is often available in handbooks.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Command Finder	**Interface Resistance** 
Simulation Navigator	Right-click the **Simulation Objects** node→**New Simulation Object**→**Interface Resistance**
Location in dialog box	**Type** list→**Edge Interface**→**Magnitude** group→**Type** list→**Heat Transfer Coefficient**

Modeling ablation and charring

What is it?

You can now model ablation and charring in the NX Multiphysics solver environment and apply the created modeling object to the NX Multiphysics solution.

The ablation or charring of a material occurs at the phase change temperature when an object is exposed to an intense transient heat source.

In addition to the thermal properties, you must define the phase change temperature and the latent heat for the material that undergoes the ablation or charring.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
	Axisymmetric Thermal	Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Creating the **Ablation-Charring** modeling object

Application	Advanced Simulation
Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Ablation-Charring**→**Create**

Applying the created modeling object to the NX Multiphysics solution

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Solution Control** node→**Thermal Control** group→**Create Ablation-Charring Material**

Appending new mapped results to a results file

What is it?

You can now append new mapped results to a BUN results file. The specified destination BUN file must have results at the same time points that you specify in the current mapping solution. To avoid overwriting results in the destination BUN file, in your mapping solution, exclude elements that already have results.

Why should I use it?

Use this option to include mapped results on a region of your model that you excluded from a previous mapping.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Multiphysics	Mapping	Thermal

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Mapping Details** node→**Data Source and Destination** group→**Mapped Data Destination** list→**Append to Selected Results File**

Aligning source and target models for mapping

What is it?

The mapping solver now can automatically align source and target models if both models have the following three named points defined: **MAP1**, **MAP2**, and **MAP3**. The three points in each model must correspond to each other. Before it maps quantities, the mapping solver automatically transforms the source model to line up the mapping points with those in the target model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Mapping	Thermal-Flow
NX Space Systems Thermal	Mapping	Thermal
NX Thermal and Flow	Mapping	Thermal-Flow
	Axisymmetric Mapping	Thermal
NX Multiphysics	Mapping	Thermal

Where do I find it?

Application	Advanced Simulation

Prerequisite	Corresponding **MAP1**, **MAP2**, and **MAP3** named points defined in source and target models
Command Finder	**Solve**
Simulation Navigator	Right-click a mapping solution node→**Solve**

Abaqus support enhancements

Abaqus keyword support enhancements

What is it?

This release includes support for a number of new Abaqus keywords.

Keyword	Import support	Export support	Supported parameters	For more information, see
`*BASE MOTION`	Yes	Yes	All required and optional parameters for `*MODAL DYNAMIC` analysis are supported.	Expanded support for Abaqus solution types
`*COHESIVE BEHAVIOR`	Yes	Yes	`ELIGIBILITY`, `REPEATED CONTACT`, `TYPE`	Cohesive behavior now supported
`*CONNECTOR BEHAVIOR`	Yes	Yes	All required parameters supported.	Element support enhancements
`*CONNECTOR DAMPING`	Yes	Yes	`COMPONENT`	Element support enhancements
`*CONNECTOR ELASTICITY`	Yes	Yes	`COMPONENT`	Element support enhancements
`*CONNECTOR SECTION`	Yes	Yes	`ELSET`, `BEHAVIOR`	Element support enhancements
`*DAMAGE EVOLOTION`	Yes	Yes	`TYPE`, `MIXED MODE BEHAVIOR`, `MODE MIX RATIO`, `POWER`, `SOFTENING`	Cohesive behavior now supported
`*DAMAGE INITIATION`	Yes	Yes	`CRITERION=MAXS`, `CRITERION=MAXU`, `CRITERION=QUADS`, `CRITERION=QUADU`, defining damage initiation as part of a contact property model.	Cohesive behavior now supported
`*DAMAGE STABILIZATION`	Yes	Yes	N/A	Cohesive behavior now supported

`*DAMPING CONTROLS`	Yes	Yes	All parameters supported.	Expanded support for Abaqus solution types
`*DASHPOT`	Yes	Yes	`ELSET,ORIENTATION`	Element support enhancements
`*DIRECT CYCLIC`	Yes	Yes	All optional parameters supported.	Expanded support for Abaqus solution types
`*DISTRIBUTING COUPLING`	Yes	Yes	All parameters supported.	Element support enhancements
`*DISTRIBUTION`	Yes	Yes	All required parameters supported.	Defining the material orientation of 3D elements with a field
`*DISTRIBUTION TABLE`	Yes	Yes	All parameters supported.	Defining the material orientation of 3D elements with a field
`*DYNAMIC`	Yes	Yes	All optional parameters for the general implicit integration method are supported.	Expanded support for Abaqus solution types
`*FREQUENCY`	Yes	Yes	`SIM, DAMPING PROJECTION`	Expanded support for Abaqus solution types
`*GLOBAL DAMPING`	Yes	Yes	All optional parameters supported.	Expanded support for Abaqus solution types
`*INITIAL CONDITIONS`	Yes	Yes	`TYPE(CONTACT), TYPE(TEMPERATURE)`	Cohesive behavior now supported Expanded support for Abaqus solution types
`*JOINT`	Yes	Yes	All required and optional parameters supported.	Element support enhancements
`*LOAD CASE`	Yes	N/A	`NAME`	Initial import support for Abaqus load cases
`*MEMBRANE SECTION`	Yes	Yes	All parameters supported.	Element support enhancements
`*MODAL DAMPING`	Yes	Yes	All parameters supported.	Expanded support for Abaqus solution types
`*MODAL DYNAMIC`	Yes	Yes	All optional parameters supported.	Expanded support for Abaqus solution types

*MONITOR	Yes	Yes	All required and optional parameters supported.	Monitoring nodes to determine solution progress
*PLASTIC	Yes	Yes	HARDENING=KINEMATIC and HARDENING=COMBINED	*PLASTIC is used in direct cyclic analysis. Expanded support for Abaqus solution types
*RESTART	Yes	Yes	All parameters supported except CYCLE and ITERATION.	Restarting Abaqus analyses in NX
*SECTION CONTROLS	Yes	Yes	HOURGLASS	Non-default hourglass control
*SELECT EIGENMODES	Yes	Yes	All parameters supported.	Expanded support for Abaqus solution types
*SOLID SECTION	Yes	Yes	REF NODE is now supported for generalized plane strain elements.	Element support enhancements
*SOLUTION TECHNIQUE	Yes	Yes	All parameters supported.	Expanded support for Abaqus solution types
*SURFACE SECTION	Yes	Yes	ELSET	Element support enhancements
*TIME POINTS	Yes	Yes	NAME, GENERATE	The *TIME POINTS keyword is used in direct cyclic analysis. Expanded support for Abaqus solution types
*VISCO	Yes	Yes	All parameters supported.	Expanded support for Abaqus solution types

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver
Menu	**File→Import→Simulation** **File→Export→Simulation**

Element support enhancements

What is it?

This release includes support for a number of additional Abaqus element types, which include:

- Distributing coupling elements
- Connector elements
- 2D diffusive heat transfer elements
- Surface elements
- Dashpot elements
- Axisymmetric shell and membrane elements
- Generalized plane strain elements
- Rigid elements

Distributing coupling elements

You can now create Abaqus **DCOUP2D** distributing coupling elements in NX. You can use a distributing coupling element to distribute forces and moments on a reference node to a collection of nodes. **DCOUP2D** elements have a number of different applications. For example, you can use a **DCOUP2D** element to do the following:

- Distribute mass to a collection of nodes.
- Distribute forces and moments on a reference node to a collection of nodes.

Abaqus element	Description	NX command for element creation	NX Physical property table
DCOUP2D	Two-dimensional distributing coupling element	**1D Connection** with the **Node to Node**, **Point to Point**, **Point to Edge**, or **Point to Face** types	N/A

Use the **Mesh Associated Data** dialog box to specify the mass to distribute to the coupling nodes.

Connector elements

You can now create Abaqus **CONN3D2** connector elements in NX. You can use connector elements when you need to connect two rigid bodies. Multi-point constraints cannot be used to connect the bodies at nodes other than the reference nodes, since multi-point constraints use degree-of-freedom elimination, and the other nodes on a rigid body do not have independent degrees of freedom.

Abaqus element	Description	NX command for element creation	NX Physical property table
CONN3D2	3-node linear element	**1D Mesh** or **1D Connection**	**Connector Section**

You can use the new **Connector Section** physical property table dialog box to define properties for the elements, such as the connection type and orientation of the connector elements. The options in the **Connector Section** dialog box correspond to the Abaqus `*CONNECTOR SECTION` keyword.

Diffusive heat transfer elements

The following table details the Abaqus diffusive heat transfer elements that are now supported in NX. You can use these elements in heat transfer analysis. Diffusive heat transfer elements allow for heat storage (specific heat and latent heat effects) and heat conduction. They also provide temperature output that you can use as input to the equivalent stress elements.

Abaqus element	Description	NX command for element creation	NX Physical property table
DC2D3	3-node linear element	**2D Mesh**	**Solid 2D Plane**
DC2D4	4-node linear element	**2D Mesh**	**Solid 2D Plane**
DC2D6	6-node quadratic element	**2D Mesh**	**Solid 2D Plane**
DC2D8	8-node bi-quadratic element	**2D Mesh**	**Solid 2D Plane**

You can use the **Solid 2D Plane** physical property table dialog box to define properties for the elements, such as material orientation and planar thermal thickness. The options in the dialog box correspond to the Abaqus `*SOLID SECTION` keyword.

Surface elements

You can now create certain types of Abaqus surface elements in NX. Surface elements are similar to Abaqus membrane elements. They are surfaces in space with no inherent stiffness. Surface elements are useful in a number of specific modeling cases. For example, you can use surface elements to bring additional mass into the model in the form of a mass per unit area. You can also use surface elements to distribute the mass of fuel in a tank over the tank surface, particularly when the tank is modeled with solid elements.

Abaqus element	Description	NX command for element creation	NX Physical property table
SFM3D3	3-node triangle element	**2D Mesh**	**Surface Section**
SFM3D4	4-node quadrilateral element	**2D Mesh**	**Surface Section**
SFM3D4R	4-node quadrilateral element, reduced integration	**2D Mesh** Use the **Element Formulation** option in the **Mesh Associated Data** dialog box to select the reduced formulation.	**Surface Section**
SFM3D6	6-node triangle element	**2D Mesh**	**Surface Section**
SFM3D8	8-node quadrilateral element	**2D Mesh**	**Surface Section**

Abaqus element	Description	NX command for element creation	NX Physical property table
SFM3D8R	8-node quadrilateral element, reduced integration	**2D Mesh** Use the **Element Formulation** option in the **Mesh Associated Data** dialog box to select the reduced formulation.	**Surface Section**

The options in the **Surface Section** physical property table dialog box correspond to the options for the `*SURFACE SECTION` keyword.

Dashpot elements

You can now create Abaqus dashpot elements in NX. Dashpot elements can be linear or nonlinear and can couple a force with relative velocity or a moment with angular velocity. You can use dashpots to model a relative velocity-dependent force or torsional resistance. You can also use dashpot elements as a mechanism for dissipating on viscous energy dissipation.

Abaqus element	Description	NX command for element creation	NX Physical property table
DASHPOT1	Dashpot element between a node and the ground that acts in a fixed direction.	**0D Mesh**	**Grounded Dashpot**
DASHPOTA	Axial dashpot element between two nodes, whose line of action is the line that joins the two nodes.	**1D Mesh** or **1D Connection**	**DashpotA**
DASHPOT2	Dashpot element between two nodes that acts in a fixed direction.	**1D Mesh** or **1D Connection**	**Dashpot**

The options in the **Grounded Dashpot**, **DashpotA**, and **Dashpot** physical property table dialog boxes correspond to the Abaqus `*DASHPOT` keyword.

Axisymmetric shell and membrane elements

You can now create Abaqus axisymmetric shell (**SAX1**, and **DSAX1**) and membrane (**MAX1**) elements in NX. You can use these elements to model axisymmetric geometry with axisymmetric deformation.

- Use **SAX1** elements in stress and displacement analyses.
- Use **DSAX1** elements in heat transfer analyses.
- Use **MAX1** elements to model thin surfaces about a symmetry axis. **MAX1** elements do not allow for any twist about that axis.

Abaqus element	Description	NX command for element creation	NX Physical property table
SAX1	2-node thin or thick linear shell, stress/displacement element	**1D Mesh** or **1D Connection**	**Axisymmetric Shell Section**
DSAX1	2-node shell heat transfer element	**1D Mesh** or **1D Connection**	**Axisymmetric Shell Section**
MAX1	2-node linear membrane element, with no twist	**1D Mesh** or **1D Connection**	**Axisymmetric Membrane Section**

The options in the **Axisymmetric Shell Section** physical property table dialog box correspond to the options and data lines for the Abaqus `*SHELL SECTION` keyword.

The options in the **Axisymmetric Membrane Section** physical property table dialog box correspond to the options and data lines for the Abaqus `*MEMBRANE SECTION` keyword.

Generalized plane strain elements

You can now create a number of different types of Abaqus generalized plane strain elements in NX. You can use generalized plane strain elements to model cases in which the structure has constant curvature with respect to the axial direction of the model. Generalized plane strain elements are typically used to model a section of a long structure that is free to expand axially or is subjected to axial loading.

The options in the **Generalized Plane Strain** physical property table dialog boxes correspond to the options and data lines for the Abaqus `*SOLID SECTION` keyword.

Abaqus element	Description	NX command for element creation	NX Physical property table
CPEG3	3-node linear triangle element	**2D Mesh**	**Generalized Plane Strain**
CPEG3H	3-node linear triangle, hybrid element with constant pressure	**2D Mesh**	**Generalized Plane Strain**
CPEG4	4-node bilinear quadrilateral element	**2D Mesh**	**Generalized Plane Strain**
CPEG4H	4-node bilinear quadrilateral, hybrid element with constant pressure	**2D Mesh**	**Generalized Plane Strain**
CPEG4I	4-node bilinear quadrilateral element, incompatible modes	**2D Mesh**	**Generalized Plane Strain**
CPEG4IH	4-node bilinear quadrilateral element, incompatible modes, hybrid with linear pressure	**2D Mesh**	**Generalized Plane Strain**

Abaqus element	Description	NX command for element creation	NX Physical property table
CPEG4R	4-node bilinear quadrilateral element, reduced integration with hourglass control	**2D Mesh**	**Generalized Plane Strain**
CPEG4RH	4-node bilinear quadrilateral element, reduced integration with hourglass control, hybrid with constant pressure	**2D Mesh**	**Generalized Plane Strain**
CPEG6	6-node quadratic triangle element	**2D Mesh**	**Generalized Plane Strain**
CPEG6H	6-node quadratic triangle element, hybrid with linear pressure	**2D Mesh**	**Generalized Plane Strain**
CPEG6M	6-node modified element, with hourglass control	**2D Mesh**	**Generalized Plane Strain**
CPEG6MH	6-node modified element, with hourglass control, hybrid with linear pressure	**2D Mesh**	**Generalized Plane Strain**
CPEG8	8-node biquadratic quadrilateral element	**2D Mesh**	**Generalized Plane Strain**
CPEG8H	8-node biquadratic quadrilateral element, hybrid with linear pressure	**2D Mesh**	**Generalized Plane Strain**
CPEG8R	8-node biquadratic quadrilateral element, reduced integration	**2D Mesh**	**Generalized Plane Strain**
CPEG8RH	8-node biquadratic quadrilateral element, reduced integration, hybrid with linear pressure	**2D Mesh**	**Generalized Plane Strain**

Rigid elements

You can now create a number of different types of Abaqus rigid elements in NX. You can use rigid elements to define the surfaces of rigid bodies for contact or for multi-body dynamic simulations. You can also attach rigid elements to deformable elements. For example, you can:

- Use **R2D2** elements in plane stress or plane strain analyses.
- Use **RAX2** elements to model axisymmetric planar geometry.
- Use **RB2D2** and **RB3D2** elements as rigid links between nodes on deformable bodies.

Rigid elements are also associated with a given rigid body and share a common node, known as the *rigid bod reference node*. In NX, use the **Rigid Body Reference Node** option in the **Mesh Associated Data** dialog box to select the appropriate reference node.

Abaqus element	Description	NX command for element creation	NX Physical property table
R2D2	2-node linear link element	**1D Mesh** or **1D Connection**	**Rigid Body Section**
R3D3	3-node, triangular facet element	**2D Mesh**	**Rigid Planar Section**
R3D4	4-node, bilinear quadrilateral element	**2D Mesh**	**Rigid Planar Section**
RAX2	2-node shell heat transfer element	**1D Mesh** or **1D Connection**	**Rigid Body Section**
RB2D2	2-node thin or thick linear shell, stress/displacement element	**1D Mesh** or **1D Connection**	**Rigid Body Section**
RB3D2	2-node rigid beam element	**1D Mesh** or **1D Connection**	**Rigid Beam Section**

The options in the new rigid body physical properties dialog boxes as well as the **Reference Node** option in the **Mesh Associated Data** dialog box correspond to the options and data lines for the Abaqus `*RIGID BODY` keyword.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part Abaqus as the specified solver

Expanded support for Abaqus solution types

What is it?

This release includes significantly enhanced support for the types of Abaqus solutions that you can create within NX Advanced Simulation. You can now perform the following types of analyses:

- Visco analyses
- Transient modal dynamic analyses
- Implicit dynamic stress and displacement analyses
- Direct cyclic analyses
- Transient heat analysis in axisymmetric thermal analyses

Visco analyses

You can now create **Visco** type steps in an Abaqus **General Analysis** solution to calculate a transient static response in a solution that includes time-dependent material behavior. Time-dependent material behavior includes:

- Creep
- Swelling
- Viscoelasticity

For example, you can create a **Visco** step to analyze quasi-static creep and swelling problems, which occur over fairly long periods of time. You can also use a **Visco** step to analyze both viscoelastic and two-layer viscoplastic materials.

Note

You can use all material models that are valid in a static analysis procedure in a **Visco** analysis step.

Use the options on the new **Visco Step Setup** tab in the **Solution Step** dialog box to specify options to control the quasi-static step as well as optional creep and damping parameters. For example, you can specify the following:

- Options to control time incrementation.
- The method to control explicit creep integration.
- Whether to have Abaqus automatically stabilize minor, local instabilities.

The options on the **Visco Step Setup** tab correspond to the parameters for the Abaqus `*VISCO` keyword.

Transient modal dynamic analyses

Use the new **Transient Modal Dynamic Step** type to analyze a transient linear dynamic problem using modal superposition. In a transient modal dynamic analysis, Abaqus calculates the response of the model as a function of time based upon time-dependent loading. The structure's response is based on a subset of the modes of the system. You must first extract these modes using an eigenfrequency extraction procedure.

You can either use the modes that Abaqus extracted in a prior eigenfrequency step or you can specify different modes.

The options on the **Transient Modal Dynamic Step Setup** tab correspond to the parameters for the Abaqus `*MODAL DYNAMIC` keyword.

Implicit dynamic stress and displacement analyses

Use the new **Implicit Dynamic** type of **Solution Step** to perform Abaqus implicit, direct integration analyses in NX. You can use an **Implicit Dynamic** step to perform:

- Dynamic stress and displacement analysis using explicit integration.

- Adiabatic stress analysis.

In these analyses, Abaqus uses implicit time integration to calculate the transient dynamic response of a system. You can use implicit direct integration dynamic analysis to study nonlinear transient dynamic response.

Use the options on the new **Implicit Dynamic Step Setup** tab in the **Solution Step** dialog box to specify the options for the analysis. For example, you can specify the following:

- The time incrementation for the dynamic stress and/or displacement analysis.
- The time integration method to use.
- The time incrementation scheme to use when contact impacts or releases occur during the analysis.

The options on the **Implicit Dynamic Step Setup** tab correspond to the parameters for the Abaqus `*DYNAMIC` keyword.

Direct cyclic analyses

Use the new **Direct Cyclic** type of step in the **Solution Step** dialog box to perform a direct cyclic procedure for nonlinear, non-isothermal, quasi-static analyses. You can use a **Direct Cyclic** analysis to predict the following:

- Progressive damage and failure for ductile bulk materials.
- Delamination or debonding growth at the interfaces in laminate composites in low-cycle fatigue analyses.

Use the options on the **Cyclic Step Setup** tab in the **Solution Step** dialog box to control, for example:

- Time incrementation.
- The number and increment of the Fourier terms.
- The time integration accuracy of the creep integration.

Additionally, you can use the new **Time Points** modeling object to specify the time points at which Abaqus evaluates the structural response.

The options on the **Cyclic Step Setup** tab correspond to the parameters for the Abaqus `*DIRECT CYCLIC` keyword. The options in the **Time Points** dialog box correspond to the parameters for the Abaqus `*TIME POINTS` keyword.

Transient heat analysis in axisymmetric thermal analyses

When you create a **Heat Transfer** solution in Abaqus **Axisymmetric Thermal** analyses, you can now create both steady-state and transient heat transfer steps. In previous releases, only steady-state heat transfer analysis was supported in the axisymmetric environment.

- Create a **Steady State Analysis** step to omit the specific heat term in the governing heat transfer equation.

- Create a **Transient Analysis** step to perform a transient heat transfer analysis. In transient steps, Abaqus performs time integration with the backward Euler method in the pure conduction elements. You can use a **Transient Analysis** step to control uncoupled heat transfer with a transient response analysis.

You can use this transient capability to model solid body heat conduction with general, temperature-dependent conductivity, internal energy (including latent heat effects), and both general convection and radiation boundary conditions.

Note that:

- If you do not include specific heat data in your material (`*SPECIFIC HEAT` keyword), Abaqus issues an error message when you solve the solution.

- If the Iterative solver method is used, you might not be able to generate numerical results if your hardware does not support multiprocessing or your Abaqus solver license does not allow you to run parallel execution. If that is the case, you will get an error message.

Additionally, in the **Solution** dialog box, you can use the new **Initial Conditions** option to specify an **Initial Temperature** value for the model. In previous releases, this value was hardcoded by the software, and you could not modify it.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver
Simulation Navigator	Right-click a Simulation→**New Solution**

Initial import support for Abaqus load cases

What is it?

NX now provides initial import support for the Abaqus `*LOAD CASE` and `*END LOAD CASE` keywords. In Abaqus, a load case refers to a set of loads, boundary conditions, and base motions that define a particular loading condition. For example, in a simplified model, the operational environment of an airplane might be broken into five load cases: (1) take-off, (2) climb, (3) cruise, (4) descent, and (5) landing.

You can use this initial support to import load cases that are associated with static perturbation steps into NX.

- If the input file contains a single load case in one solution step, NX imports all loads and constraints in the load case and associates them with the solution step in the **Simulation Navigator**.

- If the input file contains multiple load cases in one solution step, NX imports all loads and constraints in the load cases. Each imported load or constraint is named after the associated load case and step number, such as `Boundary3_in_LoadCase2_Step5`.
  - o Only loads and constraints in the first load case of the step are associated with the solution step in the **Simulation Navigator**.

- o NX places the loads and constraints from all subsequent load cases in either the **Load Container** or the **Constraint Container** in the **Simulation Navigator**. You can then add these loads and constraints to the appropriate solutions or solution steps.

For example, you import an Abaqus input file that contains two static perturbation steps, and each step has two load cases:

- Solution Step 1: `LoadCaseA` and `LoadCaseB`
- Solution Step 2: `LoadCaseC` and `LoadCaseD`

During the import process, NX creates two steps in the **Simulation Navigator**. The loads and constraints associated with load cases A, B, C, and D are imported as follows:

- All loads and constraints from load case A and C are imported and associated with steps 1 and 2, respectively. This occurs because load case A and load case C are the first load case in each step.
- All loads and constraints from load cases B and D are imported and placed in the **Load Container** or the **Constraint Container** in the **Simulation Navigator**.

In previous releases, NX ignored any `*LOAD CASE` and `*END LOAD CASE` keywords in an imported input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver
Menu	**File→Import→Simulation**

Improved support for importing Abaqus assemblies

What is it?

In Abaqus, you can define a finite element model as an assembly of individual part instances. Assemblies allow you to reuse part instance definitions, which is particularly helpful when creating large, complex models.

In NX 9, NX offered initial support for a number of the Abaqus assembly keywords, such as `*PART/*END PART`, `*ASSEMBLY/*END ASSEMBLY`, and `*INSTANCE/*END INSTANCE`. NX now imports load, constraint, and solution step data associated with part instances in an Abaqus assembly model.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver
Menu	**File→Import→Simulation**

Monitoring nodes to determine solution progress

What is it?

In Abaqus, you can monitor the progress of an overall solution through a selected node or degree-of-freedom. Use the new **Monitor** modeling object dialog box to:

- Select a specific node and degree-of-freedom to monitor.

  Note

  The degree-of-freedom you select must be valid for the selected node. For example, if the node you select belongs to a membrane element, the only active degrees-of-freedom are 1, 2, 3. If the degree-of-freedom you select is not valid, Abaqus issues an error message and the solve does not run.

- Control the frequency with which the software outputs the status of the node or degree-of-freedom.

After you create a **Monitor** modeling object, you can use the new **Monitor Request** options in the **Step** dialog box to associate the appropriate modeling object with the step.

You can change the node or degree-of-freedom that you are monitoring in individual steps by creating multiple **Monitor** modeling objects.

When you solve the solution, Abaqus prints information about the node you are monitoring in the Abaqus status file (*job-name.sta*). The status file is a text file that contains information about the progress of an analysis. You can examine the status file as Abaqus solves the solution.

The options in the **Monitor** dialog box correspond to the options for the Abaqus `*MONITOR` keyword.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver
Command Finder	**Modeling Objects**

Non-default hourglass control

What is it?

You can use the new **Hourglass Control** options in the **Mesh Associated Data** dialog box to select a non-default hourglass control approach for:

- Shell elements.
- Membrane elements (S4R and M3D4R elements).

Use the **Hourglass Control** options to select the appropriate method for hourglass control.

- Select the **Stiffness** method to base the hourglass control on the assumed enhanced strain method. You can then enter separate scaling factors for the hourglass stiffness to use with the element's displacement and rotational degrees-of-freedom.

- Select the **Enhanced** method to define hourglass control that is based on the assumed enhanced strain method. This formulation provides improved coarse mesh accuracy with slightly higher computational cost and performs better for nonlinear material response at high strain levels when compared with the default total stiffness formulation.

The new **Hourglass Control** options correspond to parameters for the Abaqus `*SECTION CONTROLS` keyword.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part Abaqus as the specified solver
Command Finder	**Mesh Associated Data**

Restarting Abaqus analyses in NX

What is it?

You can now restart Abaqus solutions from NX. For example, you can solve a solution that uses saved data from a previous analysis of the same finite element model. For complex solutions that contain multiple steps, you want to run the analysis in stages rather than analyze all steps within a lengthy, single solve. Running an analysis in stages allows you to examine the results and confirm that the solution is proceeding as you expected before you continue with the next step.

Note

When you restart a solution, you cannot change any of the basic model definition data associated with that solution, such as the mesh or the associated materials.

You can use new options to:

- Write out the necessary information to restart the analysis at a later time.
- Read in existing restart files so that you can restart the analysis.

Files required to restart a solution

To restart an analysis, you must first request that the files required for a restart be written out when you first solve a solution. These restart files allow you to complete an analysis to a certain point in the solve and then restart and continue the analysis in a subsequent solve.

Abaqus requires the following files for a restart:

- Restart file (.res)
- Analysis database (.mdl and .stt)
- Part (.prt)
- Output database (.odb)

- Linear dynamics and substructure database (.sim)

Writing out the necessary information to later perform a restart

Use the new **Restart Output Control** options on the **Output** tab in the **Solution Step** dialog box to write out restart information for a given step. You can:

- Indicate that the restart output is the same as restart data requested in a previous step
- Use the new **Restart** modeling object to define the options that control the restart of an analysis.

You can write out restart information from the following types of Abaqus steps:

- **General**
- **Static Perturbation**
- **Visco**
- **Implicit Dynamics**

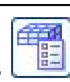
The options in the **Restart** dialog box correspond to the options for the Abaqus `*RESTART` keyword.

Reading existing restart information into a solution

To restart an analysis, you must read in the appropriate, existing restart information to your current solution. Use the options on the new **Restart** tab the **Solution** dialog box to specify the restart information to read in. For example, you can:

- Specify the name of the solution that you want to restart.
- Indicate whether you want to restart from the last available step or increment.
- Specify whether you want to resume the initial analysis or re-run the analysis from the beginning.

Where do I find it?

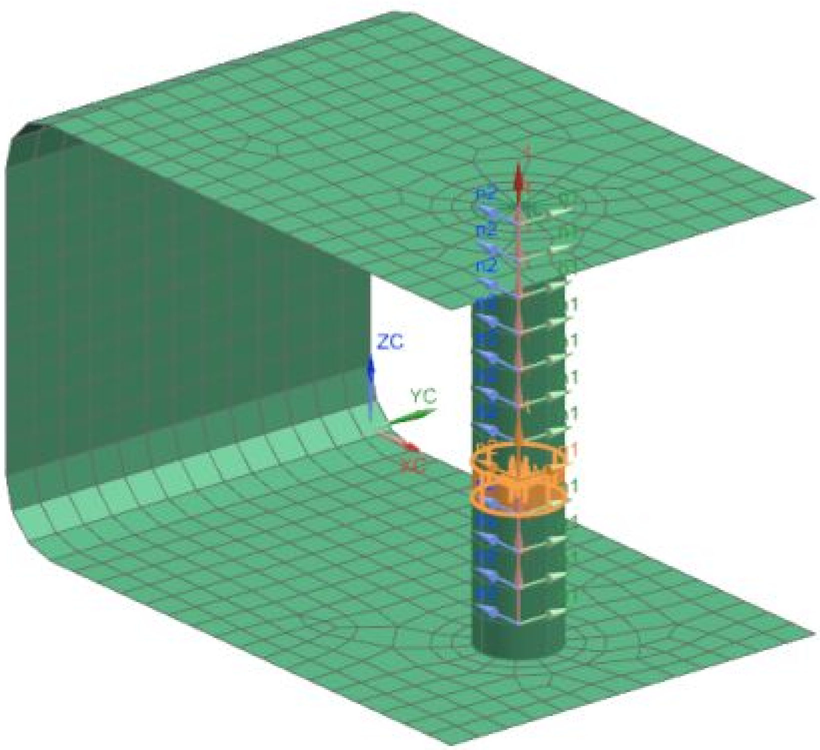
Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver **Structural** as the specified analysis type
Command Finder	**Modeling Objects** 
Location in dialog box	**Type→Restart**

Time-varying forces or adjustments for Bolt Pre-loads

What is it?

When you define a **Bolt Pre-Load** in the Abaqus environment, you can define a pre-tension load to simulate the loading of bolts in a structure. When you create a **Bolt Pre-Load**, you must specify the amount of the pre-tension force or the tightening adjustment for the bolt.

You can now use a field to define a force or tightening adjustment that varies with time for both 1D and 3D bolts. If you use a field, NX writes out a both `*CLOAD` and an `*AMPLITUDE` keyword when you export or solve the solution.



Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part Abaqus as the specified solver Structural as the specified analysis type
Command Finder	**Bolt Pre-Load**

Defining the material orientation of 3D elements with a field

What is it?

You can now define the material orientation vector of Abaqus 3D elements using a spatial field. This means that you can use the **Spatial Field** option to define the spatially varying distribution of orientation angles on the layers of Abaqus composite solid continuum elements and composite shell elements from within NX.

In previous releases, the **Spatial Field** option was available only when you imported an Abaqus input file that used the `*DISTRIBUTION` keyword to define material orientation vector data for solid elements as a field.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part Abaqus as the specified solver
Command Finder	**Mesh Associated Data**
Location in dialog box	**Material Orientation Method** list→**Spatial Field**

Cohesive behavior now supported

What is it?

When you define contact in the Abaqus environment, you can now model cohesive behavior for the contacting surfaces. In Abaqus, cohesive behavior is a generalized traction-separation behavior for surfaces. Cohesive behavior is defined as a surface-based contact property for contact pairs. Surface-based cohesive behavior is primarily intended for situations in which the interface thickness is negligibly small. For example, you can use cohesive behavior when you model the following:

- The delamination at interfaces in terms of traction versus separation.
- Two sticky surfaces that come into contact with each other during an analysis. *Sticky surfaces* are surfaces or parts of surfaces that are not initially in contact but which may bond together when they come into contact with each other. Subsequently, the bond may damage and fail.

Surface-based cohesive behavior has the following characteristics:

- It allows you to specify cohesive data, such as the fracture energy as a function of the ratio of normal to shear displacements at the interface.
- It assumes a linear elastic traction-separation law prior to damage.
- It assumes that the failure of the cohesive bond is characterized by progressive degradation of the cohesive stiffness, which is driven by a damage process.
- It allows specification of post-failure cohesive behavior if failed nodes re-enter contact.

Defining cohesive behavior for contact pairs

You can use the new **Cohesive Behavior** modeling object to define properties for the surface-based cohesive behavior. For example, you can define the traction-separation response behavior as well as specify the set of nodes affected by the cohesive behavior. After you define the **Cohesive Behavior** modeling object, select the new **Include Cohesive Behavior** check box in the **Contact Pair** dialog box to include the modeling object in the solution.

Defining damage modeling

When you include cohesive behavior in a solution, you can also include damage modeling to simulate the degradation and eventual failure of the bond between two cohesive surfaces. You specify damage behavior as part of the interaction properties for the surfaces. Cohesive surfaces can have only one damage initiation criterion and only one damage evolution law.

- Use the **Damage Initiation** modeling object to specify parameters to define the initiation of damage. Damage initiation occurs at the beginning of degradation of the cohesive response at a contact point. You can use options in the **Damage Initiation** dialog box to define the damage initiation criteria, such as the maximum separation or stresses.

- Use the **Damage Evolution** modeling object to specify the material properties that define the evolution of damage. For example, you can define the evolution of damage in terms of the energy required for failure (fracture energy) after the initiation of damage.

- Use the **Damage Stabilization** modeling object to specify the viscosity coefficients for the damage model. These coefficients are used in the viscous regularization scheme for the damage model.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver
Command Finder	**Modeling Objects**
Location in dialog box	**Type→Cohesive Behavior**

Support for nodal temperature output

What is it?

You can use the new **Structural Output Requests** dialog box to report nodal temperature data in a structural solution. On the **Thermal** tab, select **NT-Nodal Temperature**. This option corresponds to the Abaqus `NT` output variable which outputs all temperature values at a given node.

- For continuum elements, there is only one temperature value per node (NT11).

- For shell and beam elements, there are multiple temperature values per node (NT11, NT12, and so on).

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver
Command Finder	**Modeling Objects** 
Location in dialog box	**Type→Abaqus Structural Output Requests**

Specifying temperature units on import and export

What is it?

In this release, you can control the units used for temperatures in your model during import and export. This gives you the flexibility to define temperatures in a different unit system, such as Kelvin, than the rest of your model, which may be in metric or British units. In the **Import Simulation** or **Export Simulation** dialog box, use the new **Specify temperature units** list to select the unit system to use for temperatures in the model.

In previous releases, for example, if the imported unit system for your model was British units, NX imported all temperature values in the input file as °F. Similarly, if the imported unit system was metric, all temperature values were interpreted as °C.

Where do I find it?

Import Simulation dialog box

Application	Advanced Simulation
Prerequisite	A Simulation file is the displayed part and the work part Abaqus is the specified solver
Menu	**File→Import→Simulation** **File→Export→Simulation**

Improvements for appending and merging Abaqus data

What is it?

This release contains improvements when you use the **Append Merge** option in the **Solver Deck Append** dialog box to import data. Now, if your source Abaqus input file has a different analysis type or a different setting from the destination NX FEM or Simulation file for the **2D Solid Option**, NX now imports, appends, and merges the data. In previous releases, depending on the combination of the type of analysis and the **2D Solid Option** setting in the source and destination files, NX often could not import and append the data.

Where do I find it?

Application	Advanced Simulation

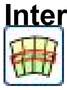
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver
Menu	**File→Append→Simulation**

Abaqus axisymmetric contact analysis (10.0.1)

What is it?

This release includes expanded support for contact analyses in the Abaqus environment. You can now perform a number of different types of contact analyses in axisymmetric structural and thermal models. In previous releases, contact analyses in the Abaqus environment were limited to non-axisymmetric models.

The following table summarizes the commands that are now available for axisymmetric structural and thermal solutions:

Abaqus element	Description	Analysis types supported	Corresponding Abaqus keywords
Edge-to-Edge Contact 	Defines contact conditions between selected polygon edges or element edges.	Axisymmetric structural	`*CONTACT PAIR` `*SURFACE`
Contact Interference 	Defines time-dependent allowable interferences for contact pairs and elements. You can use **Contact Interference** to model interference fits (press fits) when there are large overclosures between the contacting surfaces in the initial configuration of the model. The defined contact interferences can result in stresses and strains as the solver resolves the overclosure between the contacting surfaces.	Axisymmetric structural	`*CONTACT PAIR` `*CONTACT INTERFERENCE`

Abaqus element	Description	Analysis types supported	Corresponding Abaqus keywords
Contact with Clearance	Defines precise clearance or overclosure values for the nodes on the slave (target) surface in a contact pair. An overclosure value allows you to specify any initial penetration for the contacting surfaces.	Axisymmetric structural	`*CONTACT PAIR` `*CLEARANCE`
Surface-to-Surface Thermal Conductance	Models conductive heat transfer between proximate or contacting surfaces. You can model the conductive heat transfer as a function of: • The clearance between the contacting surfaces. • The contact pressure at the interface between the contacting surfaces. • Both the clearance and the contact pressure.	Axisymmetric thermal	`*GAP CONDUCTANCE`

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver Axisymmetric Structural or Axisymmetric Thermal as the analysis type General Analysis or Heat Transfer as the solution type

ANSYS support enhancements

Support for ANSYS rigid bodies

What is it?

In ANSYS, you can use rigid bodies to simulate multi-body dynamic analyses. A rigid body consists of a set of *rigid body* nodes, defined by contact elements, and a single *pilot* node, defined by a target element. The motion of the rigid body is controlled by the degrees-of-freedom at the pilot node.

Beginning in this release, you can use new options in the **Region** dialog box to define rigid bodies.

Specifying a region type and a pilot node

In the **Region** dialog box, when you select **No** from the **Use ESURF** list, you can use the options in the **Region Type and Pilot Node** group to define the rigid body.

- Use the **Region Type** list to indicate whether the region is rigid or flexible.
- Use the **Select Pilot Node** option to designate a node to serve as the pilot node.

In ANSYS, a rigid body cannot exist by itself. It must be in contact with or be connected to other bodies through contact elements. The requirements for defining a rigid body differ depending upon whether your solution is static or dynamic.

- In a static solution, you must create one **Structural Contact** definition that contains a **Source Region** and a **Target Region**.
    - o The **Source Region** must be a **Flexible** type of region.
    - o The **Target Region** must be a **Rigid** type of region and contain a specified **Pilot Node**.
- In a dynamic solution, you should generally create two **Structural Contact** definitions, although only one is required by ANSYS.

Specifying the KEYOPTs for the target elements

New **TARGE170 KEYOPTs** options have been added to the **CONTA174ET** modeling object dialog box. Use these options to specify options for the target elements, such as whether ANSYS should automatically constrain the target nodes.

Where do I find it?

Simulation Region

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Structural as the specified analysis type
Command Finder	**Simulation Region** 

CONTA174ET modeling object

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Structural as the specified analysis type
Command Finder	**Modeling Objects**
Location in dialog box	**Type**→ list **CONTA174ET**

Update of ANSYS element offerings

What is it?

In recent ANSYS releases, a number of elements have become obsolete and have been replaced by updated, enhanced versions. In this release, a number of the elements available in the ANSYS solver environment in NX have been updated to reflect those changes.

Obsolete ANSYS element	Obsolete element removed from NX?	Replacement element now available in NX
`SOLID92`	Yes	**SOLID187**
`SOLID45`	Yes	**SOLID185**
`SOLID95`	Yes	**SOLID186**
`BEAM4/BEAM44`	No	**BEAM188**
`LINK8/LINK10`	No	**LINK180**
`SHELL57`	Yes	**SHELL131**
`SHELL63`	Yes	**SHELL181**
`SHELL91`	Yes	**SHELL132/SHELL281**
`SHELL93`	Yes	**SHELL132/SHELL281**
`SHELL99`	Yes	**SHELL132/SHELL281**
`CONTAC12/CONTAC52`	No	**CONTA178**
`PLANE42`	Yes	**PLANE182**
`PLANE82`	Yes	**PLANE183**

Migration of legacy elements

How NX imports these obsolete or legacy elements depends on whether the elements have associated `KEYOPTS` in the ANSYS input file.

- If an element has associated `KEYOPTS`, then NX imports the legacy element.
- If an element does not have associated `KEYOPTS`, then NX automatically migrates the element to the appropriate replacement element. For example, NX migrates a `SOLID45` element to a **SOLID185** element.

A new **Keyopts** option has been added to the **Mesh Associated Data** dialog box to help you migrate obsolete elements to their replacements. For a legacy element, you can use this option to assign the appropriate replacement **KEYOPTS** modeling object to the element.

New customer defaults for legacy elements

This release also includes new customer defaults that you can use to facilitate the import of older ANSYS input files that may contain these obsolete or *legacy* elements. For each type of legacy element, you can use these options to specify whether they should be imported as the corresponding replacement element type or as legacy element type.

Where do I find it?

Mesh Associated Data

Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and the work part ANSYS as the specified solver
Simulation Navigator	Right-click a mesh→**Edit Mesh Associated Data**

Customer defaults

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Simulation→ANSYS→Legacy Elements** tab

New customer defaults for ANSYS element KEYOPTS

What is it?

Use the new ANSYS **KEYOPTS** customer defaults to set the default values for the `KEYOPTS` options for the following element types:

- **BEAM188**
- **SHELL131** and **SHELL132**
- **SHELL181** and **SHELL281**
- **PLANE55**
- **PLANE77**
- **PLANE182** and **PLANE183**
- **SOLID185**
- **SOLID186**
- **SOLID187**
- **SOLID226** and **SOLID227**

In ANSYS, you use `KEYOPTS` to specify additional options for elements. In NX, you specify `KEYOPTS` for ANSYS elements in the appropriate element type **Modeling Object** dialog box. For example, for **BEAM188** elements, you use the **BEAM188ET** dialog box to set `KEYOPTS`, such as the linear interpolation scheme to use.

In previous releases, the default values for ANSYS `KEYOPTS` were always set to their ANSYS default value.

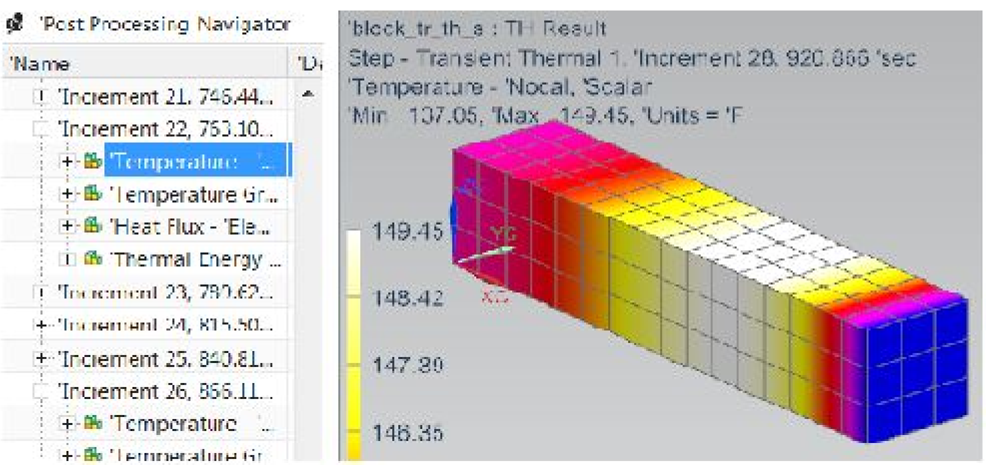
Where do I find it?

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Simulation→ANSYS→KEYOPTS** tab

Support for transient thermal analysis

What is it?

You can use the new **Transient Thermal** solution type to analyze temperatures and other thermal quantities over time. In previous releases, the ANSYS environment in NX supported only steady-state thermal analyses. Within a transient thermal analysis, you can include multiple transient steps that include time-varying (or constant) loads and constraints.



Defining an initial temperature for the solution

In a **Thermal Transient** analysis, you must first define the initial temperature value for all nodes before you analyze how that temperature changes over time. To define the initial temperature:

- If you know the initial temperature values:
  - o You can use the **Uniform Initial Temperature** option in the **Solution** dialog box to apply a uniform temperature to all nodes.

- o You can create an **Initial Temperatures** simulation object to apply different initial temperatures to selected nodes. For those nodes, the temperature value you specify in the **Initial Temperatures** dialog box overrides the value you specify in the **Uniform Initial Temperature** box.

- If you do not know the initial temperature values, you can include a **Steady State Thermal** step in your analysis immediately before the **Transient** step to calculate the temperature values.

Specifying parameters to control time increments

In addition to specifying the initial temperatures, you must also use the new **Transient Parameters** modeling object to define the time increment options.

Defining solution options

You can use options in the **Solution** dialog box to define options for the transient thermal solution. For example, you can define the following:

- Whether the loads should be ramped or stepped during the analysis, or whether the software should determine this automatically.

- Nonlinear transient solution options, which correspond to the ANSYS `THOPT` command.

- How often ANSYS outputs the solution results.

Using a transient time table

If you set the **Automatic Time Stepping** option to **On** in the **Transient Parameters** dialog box, you can create a **Transient Time Table** modeling object to define an array of key times at which the time stepping changes. Use a **Transient Time Table** when the time stepping is not constant.

The options in this dialog box correspond to the ANSYS `TSRES` array command.

Where do I find it?

Transient Thermal

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Thermal as the specified **Analysis Type** **Transient Thermal** as the specified **Solution Type**

Transient Parameters and **Transient Time Table** modeling objects

Application	Advanced Simulation

Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Thermal as the specified **Analysis Type** **Transient Thermal** as the specified **Solution Type**
Command Finder	**Modeling Objects**

Support for transient dynamic analysis

What is it?

You can now perform ANSYS transient dynamics analyses in Advanced Simulation. You use transient dynamic analysis, also called *time-history* analysis, to determine the dynamic response of a structure when time-dependent loads are applied. You can use a transient dynamic analysis to determine the time-varying displacements, strains, stresses, and forces in a structure as it responds to a combination of loads.

Note

In a transient dynamic analysis, the time scale of the loading is such that the inertia or damping effects are considered to be important.

NX supports the following methods to perform an ANSYS transient dynamic analysis:

- The full method.
- The mode superposition method.

The following table summarizes these two method.

Solution method	Description	Support for nonlinearities?
Full Method	Uses the full system matrices to calculate the transient response, with no matrix reduction.	Yes Solutions can include all types of nonlinearities, such as plasticity, large deflections, and large strains.
Mode Superposition	Sums the factored mode shapes (eigenvectors) from a modal analysis to calculate the structure's response.	No This solution can be used only for linear systems.

Where do I find it?

Application	Advanced Simulation

Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Structural as the specified **Analysis Type** **Transient Dynamic-Full Method** or **Transient Dynamic-Mode Superposition** as the specified **Solution Type**

Support for thermal-structural multiphysics analysis

What is it?

Use the new **Coupled Fields Thermal-Structural** solution type to perform coupled thermal-structural analyses in the ANSYS structural environment. NX currently supports coupled-field analysis using the direct method. The direct method typically involves one analysis that uses a coupled-field type of element that contains all necessary degrees-of-freedom. ANSYS handles the coupling by calculating element matrices or element load vectors that contain all necessary terms.

For more information, see the *ANSYS Coupled-Field Analysis Guide*.

Supported elements and properties

In this initial release, the **Coupled Fields Thermal-Structural** solution offers more complete element support for the structural portion of the solve than for the coupled portion of the solve and more limited support for the thermal portion of the solve. Currently, you can use all structural elements (0D, 1D, 2D, 3D) available in NX for the structural solve. However, you can only use SOLID226 (20-node hexadedral) or SOLID227 (10-node tetrahedral) elements to perform coupled fields analysis. These are the only types of elements that can, for example, accept both thermal and structural types of loads and boundary conditions and compute heat transfer.

Supported loads and constraints

The following thermal loads, constraints, and simulation objects have been added to the ANSYS structural environment to facilitate **Coupled Fields Thermal-Structural** analyses:

- **Thermal Constraints** and **Convection**
- **Heat Flux**, **Radiation** and **Heat Generation**
- **Automatic Thermal Coupling** and **Manual Thermal Coupling**

Note

ANSYS issues a warning message in the diagnostic file if your input file contains loads or constraints that are applied to the wrong element types. You cannot define thermal loads, constraints, and simulation objects on structural elements. You can apply thermal loads only to SOLID 226 and SOLID227 elements.

In a **Coupled Fields Thermal-Structural** solution, you use the **Structural Contact** dialog box to define both structural and thermal contact. You use the **CONTA174 Real Constants** modeling object to define the appropriate options for both structural and thermal contact in a **Coupled Fields Thermal-Structural** solution.

Specifying options for the solution and steps

When you define a **Coupled Fields Thermal-Structural** solution, use the options on the new **Multi-Fields Parameters** tab in the **Solution** dialog box to specify the following:

- Options to control the stagger iterations that the software performs. A *stagger iteration* is an iteration in which the implicit coupling of the individual physics solutions takes place. These options correspond to the ANSYS `MFITER` command.

- Options to control the relaxation values for the load transfer variables at a surface or volume interface. These options control how much of the change in a load is transferred in each stagger iteration and correspond to the ANSYS `MFRELAX` command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Structural as the specified **Analysis Type** **Coupled Thermal Fields-Structural** as the specified **Solution Type**

Data compaction options for import

What is it?

This release includes changes to how ANSYS handles extended data verification during import and new options for compacting imported data.

Extended data checking now occurs on all imported files

When you import an ANSYS input file, NX now always performs extended data checking. For example, NX checks for the existence of each node on a given element in your input file. In previous releases, you used the **Extended data checking** option in the **Import Simulation** dialog box to control whether the software performed the more comprehensive level of data validation.

New data compaction options

You can use the new **Data Compaction** options in the **Import Simulation** dialog box to access a number of options that let you control how the software writes out the ANSYS input file.

- Use the new **Import MPCs as Multi-MPCs** option to improve the import performance for models that contain a large number of multi-point constraints. When you select this option, NX imports the individual MPCs as a single **Multi-MPC** type of **Manual Coupling** constraint instead of numerous individual **CE** type of **Manual Coupling** constraints. This option offers faster import times for a large model at the expense of a more detailed representation of your input file in NX.

  This release also includes a new **Number of MPCs for Multi-MPC Creation** customer default. Use this default to specify the threshold number of multi-point constraints that a solver input file must contain before NX imports them as **Multi-MPC** couplings instead of individual **CE** couplings

- Use the **Compact displacement coordinate systems** option to improve import performance for models that contain numerous `NROTAT` commands. By default, NX creates a displacement coordinate system for every `NROTAT` command during import. This can result in numerous, duplicate displacement coordinate systems in your NX model. Select this option to have NX identify identical nodal rotations and compact them into a single displacement coordinate system. If the rotation angle between nodes is less than or equal to the specified **Material orientation comparison angle** option, NX compacts the associated displacement coordinate systems.

- Use the **Compact SHELL/PLANE element physical properties** option to improve import performance for models that have shell or planar elements with variable thickness values. Models that have varying thickness values often contain a very large number of physical property tables, which correspond to the ANSYS Real Constants and the `SECDATA` command. When you select this option, NX combines the individual property tables and stores the thickness data either as a thickness field or as element associated data.

Where do I find it?

Import Simulation dialog box

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver
Menu	**File→Import→Simulation**

Number of MPCs for Multi-MPC Creation customer default

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**ANSYS→General** tab

Specifying temperature units on import and export

What is it?

You can now control the units used for temperatures in your model during import and export. This gives you the flexibility to define temperatures in a different unit system, such as Kelvin, than the rest of your model, which may be in metric or British units. In the **Import Simulation** or **Export Simulation** dialog box, use the new **Specify temperature units** list to select the unit system to use for temperatures in the model.

In previous releases, for example, if the imported unit system for your model was British units, NX imported all temperature values in the input file as °F. Similarly, if the imported unit system was metric, all temperature values were interpreted as °C.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver

Menu	**File→Import→Simulation** **File→Export→Simulation**

ANSYS command support enhancements

What is it?

This release includes support for a number of new ANSYS commands.

Command	Description	Import support	Export support	Supported parameters
`ALPHAD`	Defines the mass matrix multiplier for damping.	Yes	Yes	`VALUE`
`ANTYPE, TRANS`	Specifies that the solution is transient.	Yes	Yes	All other parameters (`Status, LDSTEP, SUBSTEP, Action`) are not currently supported by NX.
`BETAD`	Defines the stiffness matrix multiplier for damping.	Yes	Yes	`VALUE`
`DELTIM`	Specifies the time step sizes to use for a solution step.	Yes	Yes	`DTIME, DTMIN, DTMAX, Carry` `Carry` is not currently supported in NX.
`DMPRAT`	Specifies a constant damping ratio	Yes	Yes	`Ratio,`
`IC`	Specifies initial conditions at nodes.	Yes	Yes	`NODE, Lab, VALUE, VALUE2, NEND, NINC` `NENED, NINC` are not currently supported by NX. Use the ANSYS default value for these parameters of `NENED`=1 and `NINC`=1.

KBC	Specifies stepped or ramped loading within a solution step.	Yes	Yes	KEY
LVSCALE	Scales the load vector in a **Transient Dynamic-Mode Superposition** solution.	Yes	Yes	FACT The LDSTEP parameter is not currently supported in NX. Use the ANSYS default value LDSTEP=1/
MDAMP	Defines the damping ratio as the function of a mode in a **Transient Dynamic-Mode Superposition** solution.	Yes	Yes	STLOC, V1, V2, V3, V4, V5, V6
MFITER	Specifies the number of stagger iterations for a **Coupled-Fields Thermal-Structural** solution.	Yes	Yes	MAXITER, MINITER, TARGET
MFRELAX	Specifies the relaxation values for a **Coupled-Fields Thermal-Structural** solution.	Yes	Yes	Lab, VALUE, Option
NSUBST	Specifies the number of substeps within a solution step.	Yes	Yes	NSBSTP, NSBMX, NSBMN **Note** Carry is not currently supported in NX.

THOPT	Specifies options for nonlinear **Transient Thermal** solutions.	Yes	Yes	Refopt, REFORMTOL, NTABPOINTS, TEMPMIN, TEMPMAX, --, The ALGO parameter is not currently supported in NX. Use the ANSYS default value ALGO=0.
TINTP	Defines integration parameters for transient solutions.	Yes	Yes	GAMMA, ALPHA, DELTA, THETA, OSLM, TOL, --, --, AVSMOOTH, ALPHAF, ALPHAM
TSRES	Defines an array of key times at which the time-stepping strategy changes within the solution step.	Yes	Yes	Array of key times or table of times or ERASE NX writes TSRES,ERASE before it writes a new array of key times.
TRNOPT	Specifies options for transient solutions.	Yes	Yes	Method=FULL or Method=VT only TINOPT=NMK or TINOPT=HHT for **Transient Dynamic-Full Method** solutions MSUP,,,,,NMK for **Transient Dynamic-Mode superposition** solutions. This option is not currently available in the NX user interface, but NX writes this option out when you export or solve the ANSYS input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver
Menu	**File→Import→Simulation** **File→Export→Simulation**

ANSYS axisymmetric contact analysis (10.0.1)

What is it?

This release includes support for contact analyses in ANSYS axisymmetric structural and thermal solutions. In previous releases, contact was supported only in non-axisymmetric solutions.

Axisymmetric structural solutions

You can now use the **Structural Contact** command to analyze contact between the edges of planar elements in axisymmetric models. You use the **Structural Contact** and the **Region** dialog boxes to define contact and sliding between a 2D target edge and a deformable surface. When you export or solve the solution, the software creates the appropriate CONTA172 elements to define the deformable surface and TARGE169 elements to define the target surface.

Axisymmetric thermal solutions

You can now use the **Thermal Contact** command to define thermal contact conduction between contacting surfaces in axisymmetric models using the edges of planar elements. You use the **Thermal Contact** and the **Region** dialog boxes to define the contact between the edges, as well as select the edges, points, or nodes that define the contacting edges.

After you define the contacting surfaces, use the **CONTA174/172 Real Constants** dialog box to define the required thermal contact properties. You can define the following properties:

- **Thermal contact conductance** (`TCC`), which allows you to take into account the conductive heat transfer between the contact and target surfaces.

  Note

  Currently, NX supports only a constant **Thermal contact conductance** value. You cannot vary the **Thermal contact conductance** with temperature, pressure, or time.

- The **Stefan-Boltzmann constant** (`SBCT`), which you can use to model radiative heat transfer. If you do not specify a value for the **Stefan-Boltzmann constant**, the radiation effect is excluded from the thermal contact calculations.

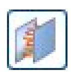
- The **Radiation view factor** (`RDVF`), which you can use to define the radiation view factor for near-field radiation.

Where do I find it?

Structural Contact

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Axisymmetric Structural as the analysis type Linear Statics or Nonlinear Statics as the solution type
Command Finder	**Structural Contact** 

Thermal Contact

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Axisymmetric Thermal as the analysis type Steady State Thermal as the solution type
Command Finder	**Thermal Contact** 

Constraint equation support in Abaqus and ANSYS thermal solutions (10.0.1)

What is it?

You can now create constraint equations between the temperature degrees-of-freedom of thermal elements in both the Abaqus and ANSYS thermal environments. In previous releases, you could only create constraint equations in the Abaqus and ANSYS structural environments.

Constraint equations for Abaqus thermal solutions

In the Abaqus thermal environment, use the new **Manual Coupling** command to define the independent and dependent coupling parameters between selected nodes. When you export or solve your model, NX creates the appropriate `*EQUATION` commands in your Abaqus input file.

Constraint equations for ANSYS thermal solutions

In the ANSYS thermal environment, the **Manual Thermal Coupling** command has been enhanced to allow you to create constraint equations as well as define sets of coupled degrees-of-freedom. Use the new **CE** option in the **Type** list of the **Manual Thermal Coupling** dialog box to define coupling between the temperature degrees-of-freedom within the model. When you export or solve your model, the software creates the appropriate `CE` keywords in your ANSYS input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus or ANSYS as the specified solver Thermal as the analysis type
Command Finder	**Manual Coupling** (Abaqus)  **Manual Thermal Coupling** (ANSYS) 

LS-DYNA support enhancements

LS-DYNA keyword support enhancements

What is it?

This release includes improved import and export support for the following LS-DYNA keywords:

Keyword	Import support	Export support	Description
`*CONTROL_BULK_VISCOSITY`	Yes	Yes	Globally resets the default values for bulk viscosity coefficients. This can be useful when modeling shock wave propagation.
`*CONTROL_OUTPUT`	Yes	Yes	Sets a variety of output parameters, such as an option to update reference node coordinates for beam elements, and the option to specify the maximum number of times a given error or warning message prints to the output file.
`*CONTROL_SHELL`	Yes	Yes	Specifies options that control the response of shell elements during an analysis.
`*CONTROL_SOLID`	Yes	Yes	Specifies options that control the response of solid elements during an analysis.
`*CONTROL_MPP_DECOMPOSITION_SHOW`	Yes	Yes	Allows the display of the final decomposition.

Keyword	Import support	Export support	Description
*CONTROL_MPP_DECOMPOSITION_TRANSFORMATION	Yes	Yes	Specifies the type of transformation to apply to the decomposition during the solution.
*DATABASE_EXTENT_BINARY	Yes	Yes	Specifies that LS-DYNA should write the output database to the D3PLOT, D3PART, and D3THDT files.
*DATABASE_BNDOUT	Yes	Yes	Specifies that LS-DYNA should create a boundary condition forces and energy output file in ASCII format.
*DATABASE_ELOUT	Yes	Yes	Specifies that LS-DYNA should create an element data output file in ASCII format.
*DATABASE_GLSTAT	Yes	Yes	Specifies that LS-DYNA should create a global data output file in ASCII format.
*DATABASE_MATSUM	Yes	Yes	Specifies that LS-DYNA should create a material energies output file in ASCII format.
*DATABASE_NODFOR	Yes	Yes	Specifies that LS-DYNA should output nodal force groups data to the nodal point data output file.
*DATABASE_RBDOUT	Yes	Yes	Specifies that LS-DYNA should create a rigid body data output file in ASCII format.
*DATABASE_RCFORC	Yes	Yes	Specifies that LS-DYNA should create a resultant interface data output file in ASCII format.
*DATABASE_SLEOUT	Yes	Yes	Specifies that LS-DYNA should create a sliding element interface output file in ASCII format.

Keyword	Import support	Export support	Description
`*PART_COMPOSITE_TSHELL`	Yes	Yes	Provides a simplified method of defining a composite material model for thick shell elements. This eliminates the need for user-defined integration rules and part IDs for each composite layer.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and work part LS-DYNA as the specified solver
Menu	**File→Import→Simulation** **File→Export→Simulation**

Access to advanced solver options

What is it?

You can use the new **Edit Advanced Solver Options** button in the **Solve** dialog box to access a number of options that let you control how the software writes out the LS-DYNA keyword file. For example, you can control the following:

- The name and location of the LS-DYNA keyword file.
- The real data filter, which controls whether all data with an absolute value smaller than the specified filter is written to the LS-DYNA keyword file as 0.0.
- Whether all data or selected types of data is exported to the LS-DYNA keyword file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and work part LS-DYNA as the specified solver
Simulation Navigator	Right-click solution→**Edit Advanced Solver Options**

Support for global damping

What is it?

Use the new **Damping** modeling object to define mass-weighted nodal damping that applies:

- Globally, to the nodes of deformable bodies.

- To the center of mass of rigid bodies.

You can use the options in the **Damping** dialog box to globally damp low frequency structural modes as well as rigid body modes. For example, you can use options to:

- Select a table-based field to define the loading curve that defines the system damping constant.
- Specify scaling factors on the global X, Y, and Z translational and rotational damping forces.

After you create a **Damping** modeling object, you can use the new **Damping** tab in the **Solution** dialog box to specify which **Damping** modeling object to include in the solution.

The options in the **Damping** dialog box correspond to the LS-DYNA `*DAMPING_GLOBAL` keyword.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and work part LS-DYNA as the specified solver
Command Finder	**Modeling Objects** 
Location in dialog box	**Type** list →**Damping**

Expanded support for *CONTROL and *DATABASE keywords

What is it?

This release includes improved support for the LS-DYNA `*CONTROL` and `*DATABASE` keywords.

Improved support for the *CONTROL keyword

This release includes expanded support in NX for the LS-DYNA `*CONTROL` keyword. The **Controls** modeling object dialog box has been updated with a number of new options. For example, you can:

- Use the options on the new **Bulk Viscosity** tab to change the default values for bulk viscosity coefficients. This can be useful when you model shock wave propagation.
- Use the options on the new **Output** tab to set a variety of output parameters for LS-DYNA. For example, you can specify options to update reference node coordinates for beam elements and to control the maximum number of times a given error or warning message prints to the output file.
- Use the options on the new **Shell** tab to specify options for shell elements. For example, you can set a maximum allowable for warp value for shell elements. If the warp value for shell elements in your model exceeds this limit, LS-DYNA prints a warning message.

For a list of all newly supported `*CONTROL` keywords, see Selectively importing and exporting LS-DYNA data.

Improved support for the *DATABASE keyword

This release also includes expanded support for the LS-DYNA `*DATABASE` keyword. The **Database** modeling object dialog box has been updated with a number of new options. For example, you can:

- Use the options on the **Extent Binary** tab to control some of the data written to the LS-DYNA binary output databases (d3plot, d3thdt, and d3part). You can, for example, choose to include output at additional integration points for shell and solid elements. You can also control whether stress or strain tensor data is written to the binary databases.

- Use the options on the **General Option** tab to control which ASCII output databases are created by LS-DYNA during the solve.

For a list of all newly supported `*DATABASE` keywords, see Selectively importing and exporting LS-DYNA data.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part LS-DYNA as the specified solver
Command Finder	**Modeling Objects**
Location in dialog box	**Type→Controls** or **Database**

Selectively importing and exporting LS-DYNA data

What is it?

You can now selectively import and export LS-DYNA data from NX.

- In the **Import Simulation** dialog box, use the options in the new **Selective Import** group to control which specific commands NX imports from the specified input file. For example, you can clear the **Materials** check box if you do not want to import any LS-DYNA materials keywords.

- In the **Export Simulation** dialog box, use the options in the new **Export Options** group to control which parts of your NX model file are exported to the LS-DYNA keyword file.

Additionally, you can use the new **Import unsupported keywords** option in the **Import Simulation** dialog box to import LS-DYNA keywords that are not currently supported in NX as either commented or uncommented user defined text. For example, you can use this new option to import LS-DYNA load and constraints keywords as uncommented user defined text. If you later solve your LS-DYNA solution from NX, LS-DYNA should solve the file and produce correct results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and work part LS-DYNA as the specified solver
Menu	**File→Import→Simulation** **File→Export→Simulation**

Merging parts of an LS-DYNA input file into an existing NX model

What is it?

You can now use options in the **Solver Deck Append** dialog box to merge entities within an existing LS-DYNA input file into an existing NX model. These options allow you to import portions of an input deck into an existing model, as long as the missing portions of the input file already exist in NX.

For example, you can:

- Import only elements from an input file into an NX FEM that already contains nodes.
- Import group definitions that reference nodes and elements where the nodes and elements already exist in NX.
- Import loads and constraints into a model that already contains a mesh.

In previous releases, you could append an LS-DYNA solver input file into an existing NX model only if the input file being appended was a complete, valid, standalone file.

Controlling import behavior

You can use the options in the **Import Behavior** list to control how NX imports and appends the data from the LS-DYNA input file:

- Select **Append** to import a solver input data file into an existing FEM or Simulation file and append the solver data to the existing data. The solver input data file that you import must be complete and valid.
- Select **Append Merge** to import data from an incomplete solver data input file into an existing FEM or Simulation file and merge that data into the existing NX file. The solver data input file that you import does not need to be complete.

  When you use the **Merge Entities** option to append data, NX does not modify any entities that were present in NX prior to the merge. For example, NX does not modify the following:

  - Solution attributes.
  - Values in existing loads, boundary conditions, simulation objects, or modeling objects.
  - The content of existing NX groups.

Ability to create a new solution

Use options in the **Create Solution** list to control whether the source LS-DYNA input file data is appended into a new solution in NX. In previous releases, the appended entities were not placed in a new solution.

- Select **No** if you do not want to create a new solution. All loads and boundary conditions are placed in the appropriate containers in the **Simulation Navigator**, but they are not referenced in the current solution. Any appended modeling objects are also not referenced in the current solution.

- Select **Yes** to create a new solution. All loads and boundary conditions are placed in the appropriate containers in the **Simulation Navigator** and referenced in the current solution. Any appended modeling objects are also referenced in the new solution.

Options for handling data conflicts

This release also includes options that let you control how NX handles conflicts between nodes, materials, and physical property data in the LS–DYNA input file and the destination FEM or Simulation file.

- Select **Ignore Deck Data** if you do not want to append an entity from the solver input file when it conflicts with an existing entity in the NX destination file. For example, if both the source solver input file and destination NX file have a coordinate system with an ID of 15, NX does not append the coordinate system from the solver input file.

- Select **Modify Using Deck Data** to replace the node, material, or physical property data in the NX destination file with the data from the source solver input file.

- Select **Append Deck Data** to specify a numerical offset to increment the label (ID) when conflicts occur. If NX detects a conflict between an entity in the source input file and the destination NX file, it uses the offset to change the ID of the entity in the source input file during the append. The software also updates any references to that entity in the source file. For example, if you append an `ELEMENT_SHELL(4)` element that originally referenced nodes 1 and 2, and those nodes are renumbered with an offset of 1000 because of a conflict, the software updates the `ELEMENT_SHELL(4)` element to reference nodes 1001 and 1002.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file as the displayed part and work part LS-DYNA as the specified solver
Menu	**File→Append**

Inflated laminates in LS-DYNA (10.0.1)

What is it?

The **Extrude Laminate** and **Fill Laminate** commands are now supported in the LS-DYNA environment.

The extruded 3D mesh consists of ELEMENT_TSHELL wedge or hexahedral elements and the ply drop-offs consist of ELEMENT_SOLID tetrahedral, wedge, or hexahedral elements. Because LS_DYNA does not support pyramids, pyramids are split into tetrahedral elements.

For both commands, NX creates a 3D mesh collector that has a **Solid Laminate** physical property assigned to it. NX automatically sets the **Stacking Recipe** in the **Solid Laminate Modeler** dialog box to **Inherited from Layup**.

All the plies of the extruded solid laminate have the same ply failure theory and inter-laminar failure theory that you selected in the **Laminate Modeler** dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM within the LS-DYNA solver environment.
Command Finder	**Extrude Laminate** or **Fill Laminate**

Samcef solver environment

Samcef solver environment

What is it?

This release provides support in NX Advanced Simulation of the new **Samcef** solver environment. When you work in the Samcef environment, you can:

- Create a Samcef model, export it, and perform a Samcef linear static, modal, buckling, or nonlinear static analysis. You can then post-process the results using NX Post-processing.
- Preview the real contact and glue conditions computed by Samcef, using the **Ancillary Display** mode.
- Specify materials such as progressive composite damage material models.
- Create cohesive elements to model the interface between two layers, define a damage interface material and analyze phenomena such as delamination.
- Import a Samcef model or post-process results from other Samcef analysis types that are not yet supported in the NX **Samcef** solver environment.

Supported element types

NX supports a selection of Samcef 0D, 1D, 2D, and 3D elements and their associated physical properties:

- 0D lumped mass, grounded bushing, and spring elements.
- 1D beam and rod elements, as well as 1D connection elements (rigid links and averaged kinematic constraints).
- 2D Mindlin thin shell, Heterosis shell, 2D membrane or shell membrane, metallic or composite.
- 3D solid and solid shell, metallic and composite elements.

  3D cohesive elements.

NX supports linear and quadratic elements, true pyramid transition (Tet to Hex transition), and mixed order transition.

NX automatically groups the elements in mesh collectors and meshes according to their types, physical properties, and materials.

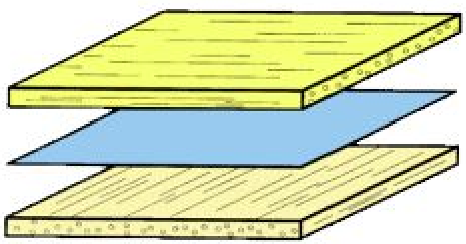
NX Laminate Composites (NXLC)

The Samcef environment is fully coupled with NX Laminate Composites, including:

- 2D and 3D laminates definitions (multi-layered elements).
- Inflation process to create 3D laminates.
- Composite materials management.

NX Laminate Composites also generates the Samcef 3D cohesive elements. You can use cohesive elements to model the interface between two layers and phenomena such as delamination.

You can also use cohesive elements to compute interlaminar stresses (glue modeling) and get accurate results between plies.



Cohesive elements are 3D elements with the same topology as classical brick and wedge elements, but they can have a zero thickness. You can create cohesive elements with the inflation process of a composite definition.

For more information, see Samcef solver support in NX Laminate Composites and *Cohesive layers* in the Advanced Simulation online Help.

Support for materials

You can use isotropic, orthotropic, and elastoplastic materials. You can also define the following progressive composite damage models for:

- General orthotropic ply (Samcef `UD Ply` damage).
- Enhanced ply (NCF) (Samcef `Enhanced UD Ply` damage).
- Woven fabrics material (Samcef `Woven Ply` damage).

For more information, see Samcef damage properties

NX also supports the damage interface material model for cohesive elements. For more information, see *Samcef Damage Interface material* in the Advanced Simulation online Help.

Supported loads and constraints

NX supports many loads, such as nodal force, pressure, speed, acceleration, and enforced temperature. You can also define constraints, such as fixation, enforced displacements and rotations, plane of symmetry, or manual coupling.

For more information, see *Boundary conditions for Samcef solutions* in the Advanced Simulation online Help.

Supported contacts and glue conditions

NX supports the following glue conditions:

- **Nodes-to-Surface** gluing.
- **Solid-to-Edge** gluing.
- **Edge-to-Edge** gluing.
- **Surface-to-Surface** gluing.
- **Edge-to-Surface** gluing.

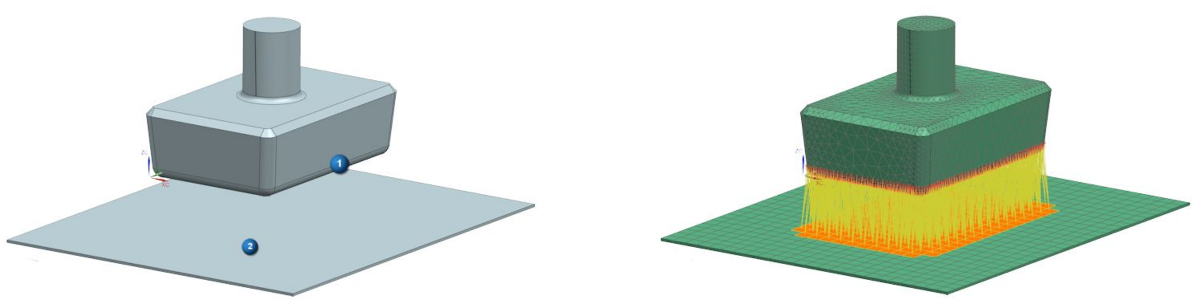
NX supports the following contact conditions:

- **Surface-to-Surface** contact.
- **Edge-to-Surface** contact.
- **Rigid-Flexible** contact.

Ancillary Display preview for contact and glue conditions

When you solve a model that contains either contact or glue conditions, Samcef internally creates contact or glue elements. You can understand where those elements are created by using the **Ancillary Display** mode to preview in NX the real contact and gluing conditions as estimated by Samcef at the initial stage.

For instance, in a contact between 2 surfaces of a forming stamp, you can preview the real contact area computed by Samcef:



For more information, see *Previewing contact and glue element locations (Samcef)* in the Advanced Simulation online Help.

Supported analysis types

When you create a Simulation file with Samcef as the specified solver, you can perform only a structural analysis.

Supported solution types

NX supports the following Samcef solution types:

Solution type	Description
Linear Static	Lets you perform a linear static solution of a structure. Samcef can take into account load cases, gluing, and small sliding contact conditions.
Modal	Lets you compute natural frequencies and vibration modes, as well as deformed shapes and the corresponding modal stresses.
Nonlinear Static	Lets you perform a nonlinear solution to analyze critical conditions such as plasticity. You can take into account gluing, large sliding contacts, and time-dependent loads. You can control the nonlinear computation by using strategy parameters such as automatic time step definition, initial static computation parameters, maximum iteration number, or energy and force thresholds. Composite models are handled as well.
Buckling	Lets you perform a buckling solution to determine buckling loads and buckling mode shapes of a linear elastic structure.

Support of User Defined Text

In the **Solution**, you can add specific text to include solver commands or options that are not yet supported. This text will be appended to your solver input file.

For more information, see *Customizing a Samcef input file with user defined text* in the Advanced Simulation online Help.

Post-processing of results

You can post-process results from a Samcef analysis directly in NX Post-processing (.des / .fac files):

- You can post-process all the classical structural results as stresses, strains, shell resultants, and beams results.
- For laminates, you can display in-ply stresses and strains, out-of-ply stresses, damage results, delamination, and native failure criteria results.
- You can display results in an absolute, material, native, or user-defined coordinate system.
- You can create XY graphs to plot results of a same type across a sequence of time steps or load cases, for example, but also plot a single result across a range of nodes or elements.
- You can also post-process results from many other Samcef analysis types, such as thermal analyses.

  Note

  To post-process Samcef results, you must activate these **Customer Defaults**:

  1. Set the **Additional Results File Extension** option to `DES`.
  2. Select the **Read Unknown Results** option.

Import of Samcef models

You can import a structural Samcef model into NX from the following Samcef files:

- ASCII formatted files (.dat), also known as Samcef `Bank` files.
- Binary formatted files database (.sdb), also known as Samcef `SDB` files.

You can import the nodes, elements, physical properties and materials. The import supports units conversion and can create groups or regions. Elements are automatically grouped in mesh collectors and meshes according to their types, physical properties, and materials.

NX does not import Samcef composite data such as the plies (.PLI) and the laminates (.LAM).

NX stores unsupported data in **User Defined Text** objects.

For more information, see *Import overview* in the Advanced Simulation online Help.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file as the work part and displayed part Samcef as the specified solver

Samcef solver support in NX Laminate Composites

What is it?

You can access most NX Laminate Composites commands and functionality in the Samcef solver environment.

All laminate commands are supported except the following:

-  **Solid Laminate Physical Property**
-  **Ply Materials**

Ply IDs in all laminate physical properties and global layups must have unique values. In other solver environments, ply IDs in different laminate physical properties can have the same values.

In the Samcef environment, you can create cohesive layers between extruded plies to model delamination. See Cohesive layers for more information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the work part and displayed part Samcef as the specified solver

Samcef support enhancements (10.0.1)

This release includes a number of improvements to the Samcef solver environment.

Creating and solving solutions

- A new **Compute curvature for Mindlin shells and beams** option has been added to the **Advanced Parameters** tab in the **Solution** dialog box for all Samcef solutions. When you select this option, Samcef introduces a curvature based on nodes into the element by computing the shell's curvature from mean nodal normals.

- In the **Advanced Solver Options** dialog box, use the new **Sorted Data (Nodes/Elements)** option to sort the nodes and elements labels in the Samcef input file when you export or solve your solution.

- On x86_64 Linux and Windows 64-bit operating systems, you can use two versions of the Samcef solver: the `i4` version, which uses 32-bit integers and is limited to 8GB of memory, and the `i8` version, which uses 64-bit integers and is limited to 16TB of memory. Samcef automatically uses the `i8` version of the solver, which improves performance on large models while remaining compatible with results from the `i4` version of the solver.

- When you restart a solution that has existing results, the process by which the software deletes existing result files and any intermediate files has been improved.

NX Laminate Composites

This release contains improved support for Samcef models in the NX Laminate Composites toolset.

- You can now use the **Ply Materials** command to assign material properties to laminate plies. This command was not supported for Samcef models in the NX 10 release.

- You can now use the **Advanced Post Report** command to quickly identify critical element and ply data over many solutions. The enveloping, filtering, and sorting tools help you to find the critical element and ply data that you want without reviewing all the solver ply data.

Post-processing

In the **Post Processing Navigator**:

- The listing and names of the Samcef stress and strain result types in the **Post Processing Navigator** are now more clear. (1) shows an example of Samcef results listed in the **Post Processing Navigator** in NX 10. (2) shows the same results listed in the NX 10.0.1 release.

- You can now load results from Samcef analyses faster than in previous releases. This improvement is most noticeable when you load results from large models and can be up to 50% faster, depending on the size of your model.

Importing Samcef data

- The time that it takes to import large Samcef input files that contain numerous **Region** definitions has been greatly reduced.

- When you import a Samcef input file that contains groups of faces and edges, NX now imports them as either a **Surface Region** or an **Edge Region**.

NX Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal enhancements

General capabilities

Solution Monitor enhancements

What is it?

- The **Solution Monitor** is now used by the NX Multiphysics and NX Nastran solvers.

- The **Solution Monitor** now displays text output for each solver on a separate tab when you run a coupled NX Multiphysics analysis.
  - o The **NX Multiphysics** tab displays the text output for the coupled solver.
  - o The **NX Thermal** tab displays the text output for the thermal solver.
  - o The **NX Nastran** tab displays the text output for the NX Nastran solver.

- You can copy and paste the displayed text output. In previous versions, the displayed text output was not selectable.
- The user interface of the **Solution Monitor** is refreshed and the command names are more intuitive. The commands now have tooltips describing their functionality.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	NX Nastran, NX Multiphysics, NX Thermal and Flow, NX Space Systems Thermal, or NX Electronic Systems Cooling as the selected solver environment
Command Finder	**Solve**
Simulation Navigator	Right-click a solution node→**Solve**

Expression and 4D field support for boundary conditions

What is it?

The thermal solver supports expressions and 4D fields defined on boundary conditions. The expressions can contain variables, fields, and functions that the thermal solver can evaluate at solve time. See Expressions for more information.

The flow solver supports only 4D fields that are defined on boundary conditions.

You can now define the magnitude of certain boundary conditions as a function of both space and another independent variable. For example, you can define a total conductance in the **Thermal Coupling** simulation object to vary in space and with temperature.

A 4D field is either a heterogeneous table or a table of fields. A heterogeneous table is one where the dependent variable is a function of both space and a non-spatial quantity, for example, time. A table of fields is constructed by defining a distinct spatial field for each value of the independent non-spatial variable.

Why should I use it?

This enhancement provides the ability to:

- Use solver-evaluated quantities when you define thermal boundary conditions. You can also define interdependencies between boundary conditions, for example, when you want to define the outlet temperature of one **Thermal Stream** load as the input temperature of another.
- Define boundary conditions that vary both in space and time simultaneously. Previously, the magnitude could vary spatially and the spatially-varying magnitude was scaled as a function of time. Now, at each node, your magnitude can vary in time independently of other nodes.

Aligning source and target models for mapping

What is it?

The mapping solver now can automatically align source and target models if both models have the following three named points defined: **MAP1**, **MAP2**, and **MAP3**. The three points in each model must correspond to each other. Before it maps quantities, the mapping solver automatically transforms the source model to line up the mapping points with those in the target model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Mapping	Thermal-Flow
NX Space Systems Thermal	Mapping	Thermal
NX Thermal and Flow	Mapping	Thermal-Flow
	Axisymmetric Mapping	Thermal
NX Multiphysics	Mapping	Thermal

Where do I find it?

Application	Advanced Simulation
Prerequisite	Corresponding **MAP1**, **MAP2**, and **MAP3** named points defined in source and target models
Command Finder	**Solve** 
Simulation Navigator	Right-click a mapping solution node→**Solve**

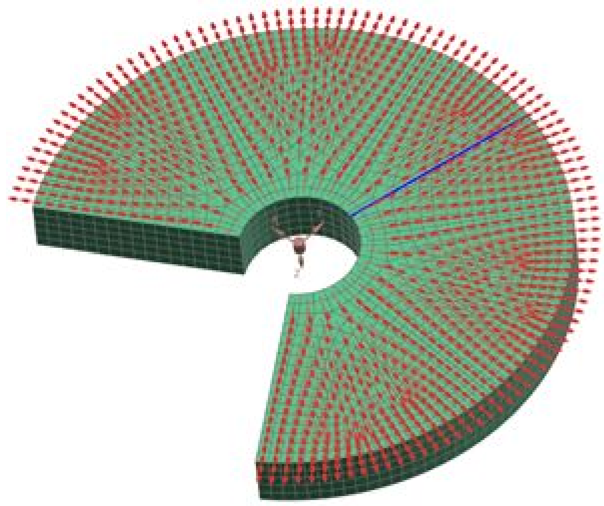
Thermal capabilities

Defining a thermal stream with a radial coordinate system

What is it?

You can define convection caused by radial thermal streams over surfaces. The stream flows radially over the surface starting from the origin of the specified cylindrical coordinate system.

The thermal solver creates a 1D duct with mass flow elements on the selected regions and, using convection thermal coupling, connects the 1D elements to the nearest thermal solid elements that lie in the same radial band. The 1D duct is shown in blue in the following figure.



You can set the radial **Thermal Stream** load type based on whether you want the thermal stream to convect to a single surface or to two surfaces.

In addition to specifying the region, the fluid material, the stream conditions, and the heat transfer, you also specify the cylindrical coordinate system. In the structural multiphysics solution, you only specify the pressure.

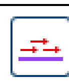
Why should I use it?

The load provides an automated way to include the effects of convection and advection in a thermal model. It improves your efficiency by regrouping many thermal simulation boundary conditions in a single command.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Structural	Multi-Step Nonlinear
	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	The specified solver must be NX Multiphysics or NX Thermal and Flow.
Command Finder	**Thermal Stream** 
Simulation Navigator	Right-click the **Load** container node→**New Load**→**Thermal Stream**
Location in dialog box	**Type** list→**One-Sided Stream on Faces (Cylindrical Components)** or **Two-Sided Stream on Faces (Cylindrical Components)**

Boundary Condition ID

What is it?

You can now insert the boundary condition ID into a boundary condition name by including the symbol `%%ID`. For example, if you create a thermal void with an ID of 6 and set the name to `Void – %%ID`, the name appears as `Void – 6`.

You can also pass the boundary condition ID as an argument of some built-in functions when you define boundary conditions using the expressions.

You can do this for any type of boundary condition that includes the boundary condition ID in its dialog box. Currently, these boundary conditions are:

- **Thermal Convecting Zone**
- **Thermal Stream**
- **Thermal Void**

Built-in functions with the boundary condition ID as an argument

In the following functions, “i” represents the boundary condition ID value.

Built-in function	**Description**
STI(i)	Returns the total absolute or relative inlet temperature of the stream i.
STO(i)	Returns the total absolute or relative outlet temperature of the stream i.
VT(i)	Returns the total absolute or relative temperature of the void i.
ZT(i)	Returns the total absolute or relative temperature of the convecting zone i.
SMO(i)	Returns the outlet mass flow of the stream i.
STMO(i)	Returns the outlet metal temperature of the stream i.
SP(i)	Returns the pressure of the stream i.
VP(i)	Returns the pressure of the void i.
ZP(i)	Returns the pressure of the convecting zone i.
SSV(i)	Returns the swirl velocity of the stream i.
VSV(i)	Returns the swirl velocity of the void i.
ZSV(i)	Returns the swirl velocity of the convecting zone i.
SA(i)	Returns the surface area of the stream i selection.
VA(i)	Returns the surface area of the void i selection.
ZA(i)	Returns the surface area of the convecting zone I selection.
MIX(i1,i2,...,i10)	Returns the temperature from mixing the flows of two or more streams, i1, i2, up to i10.
MMIX(i1,i2,...,i10)	Returns the mass flow from mixing the flows of two or more streams, i1, i2, up to i10.

Why should I use it?

You can use the boundary condition ID to define interdependencies between boundary conditions; for example, to model flow recirculation of thermal streams, you define the outlet temperature of one thermal stream as a function of the input temperature of another stream.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Structural	Multi-Step Nonlinear
	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics or NX Thermal and Flow as the specified solver.
Command Finder	**Thermal Convecting Zone** or **Thermal Stream** or **Thermal Void**
Simulation Navigator	Right-click the **Load** container node→**New Load**→**Thermal Convecting Zone** or **Thermal Stream** or **Thermal Void**
Location in dialog box	**Name** group→**Boundary Condition ID**

Solid Motion Effects simulation object enhancement

What is it?

Use the **Spinning - All Elements** type of **Solid Motion Effects** simulation object to model the thermal effects of a spinning model. All elements in the model spin together.

You select the spinning axis, and specify the number of calculations per spin, *N*. The thermal solver calculates the view factors and thermal couplings that result from NX spinning the model *N* times, each time rotating the spinning model by 360/*N* degrees. The thermal solver then averages the view factors and thermal couplings of these *N* positions.

The view factors include the following:

- Solar, earth, and albedo view factors from orbital heating.
- Heat flux view factors from radiative heating.
- Black body and ray-traced view factors from the deterministic and hemicube methods.
- Ray-traced view factors, radiative couplings, and heat loads from the Monte Carlo method.

Note

Best accuracy is obtained when *N* is greater than the number of elements around the circumference of the spinning part. This is especially true for thermal coupling calculations.

The **Spinning** type of **Solid Motion Effects** simulation object from the previous releases is now called **Spinning - Selected Elements**.

Why should I use it?

The **Spinning - All Elements** type of **Solid Motion Effects** simulation object can be used, for example, to model a spinning spacecraft that spins rapidly around a specified axis.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solid Motion Effects** 
Simulation Navigator	Right-click the **Simulation Object** container node→**New Simulation Object**→**Solid Motion Effects**
Location in dialog box	**Type** list→**Spinning - All Elements** or **Spinning - Selected Elements**

Varying capacitance for Void Non Geometric Element modeling object

What is it?

You can now define a time-varying or temperature-varying capacitance when you create a **Void Non-Geometric Element** modeling object. To define the capacitance, you can specify the capacitance value or the volume value. Both quantities can vary with time or temperature.

In previous releases, you could specify only a constant capacitance value.

Why should I use it?

You can use time-varying capacitance, for example, to model the capacitance of fuel in aircraft wings. The fuel mass and thus the capacitance varies with time over a flight profile.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

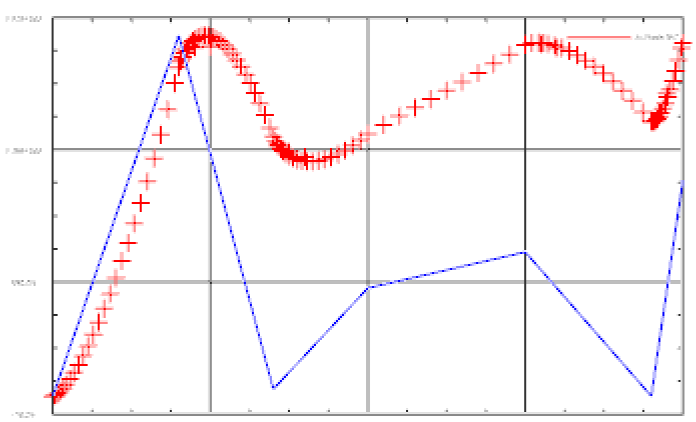
Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics or NX Thermal and Flow as the specified solver.
Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Void Non-Geometric Element**→**Create**→**Calculation Method** group

Adaptive time stepping enhancement

What is it?

The new adaptive time stepping scheme handles the sharp changes in temperature at boundary conditions better than the previous scheme. When there are no abrupt changes in boundary conditions, it accelerates the simulation without losing accuracy. The time step size calculation is now based on the estimated error between the analytical solution and the simulation. The error is the difference between a fit quadratic function and a linear variation through three consecutively computed temperature values for two consecutive time steps.



When there are abrupt temperature changes in the model, as seen in the example, the adaptive time stepping scheme adapts by performing smaller time steps around the times when the abrupt changes occur. The blue curve represents the time-varying heat load that is applied to the boundary condition and each red + represents the temperature value at one point of the boundary condition for the transient run. The +'s that are close to each other indicate that the time steps are smaller at those times to better capture the changes in the heat load.

You specify the temperature error tolerance, the minimum time step size, and the maximum time step size.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal Advanced Axisymmetric Thermal

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Transient Setup** node→**Time Integration Control** group→**Time Step Option** list→**Automatic**

Local temperature error estimate result sets

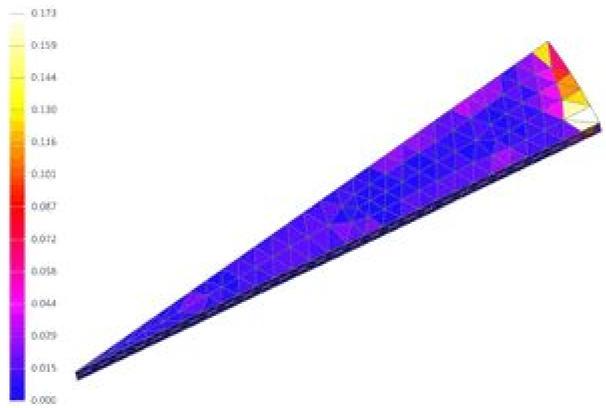
What is it?

You can now recover local temperature error estimate results. The thermal solver generates temperature error estimate results which estimate the local spatial discretization errors due to the mesh used for the simulation of conduction.

After you solve the model, the following result types appear under the solution node in the **Post Processing Navigator**:

- **Temperature Error Estimates - Nodal**
- **Temperature Error Estimates - Elemental**
- **Time of Maximum Error Estimates - Nodal**
- **Time of Maximum Error Estimates - Elemental**
- **Maximum Temp Error Estimates - Nodal**
- **Maximum Temp Error Estimates - Elemental**

The last four result types are written only for transient runs. They are displayed under the **Time Invariant** node in the **Post Processing Navigator**.



The error estimate is measured in units of temperature difference.

The elemental error estimate, ε_T, is the maximum temperature difference of all boundary elements computed as follows:

$$\varepsilon_T = \max |T_{BE} - f_{BE}(T_1, T_2, \ldots, T_N)|$$

where:

- T_{BE} are the calculated boundary element temperatures
- T_i are the N smoothed nodal temperatures
- N is the number of nodes
- The f_{BE} function uses element shape functions to interpolate the nodal temperatures to the boundary element

Why should I use it?

The adaptive meshing solution process uses the maximum local temperature error estimates to refine the mesh for conduction thermal analysis.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal Advanced Axisymmetric Thermal

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**

Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Results Options** node→**Thermal** group→ **Local Temperature Error Estimate**

Envelope results for thermal post processing

What is it?

Time invariant results from transient thermal solutions are now located under the **Time Invariant** node in the **Post Processing Navigator**.

- iron
  - Solution 1
    - Thermal
      - Increment 1, 0.0 sec
      - ...
      - Increment 61, 600.000 sec
      - Time Invariant
        - Max Temperature - Elemental
        - Min Temperature - Elemental
        - Time at Max Temperature - Elemental
        - Time at Min Temperature - Elemental

The time invariant thermal results include the following:

- Maximum and minimum elemental temperature over the complete transient run and the times at which they occur.
- Maximum thermal error estimates at each element over the complete transient run and the times at which they occur.
- Absorbed, incident, reflected, or transmitted radiative fluxes at each element when these values are constant over the complete transient run.

In previous releases, these results were displayed in the first or last time step.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal Advanced Axisymmetric Thermal
	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	Loaded results file that contains transient thermal results.

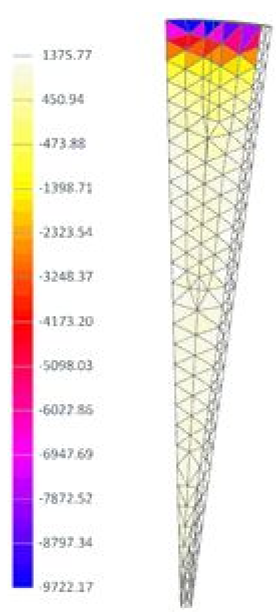
Convective heat flux results

What is it?

You can now recover convective heat flux results on 2D shell elements. You request nodal, elemental, or nodal and elemental types of results from the **Convective Heat Flux Results Location** list.

After you solve the model, depending on your request, either or both of the following result types appear under the solution node in the **Post Processing Navigator**:

- **Convective Heat Flux - Nodal**
- **Convective Heat Flux - Elemental**



Why should I use it?

Convective heat flux results help you analyze the results of your thermal simulation.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Results Options** node→**Thermal** group→ **Convective Heat Fluxes**

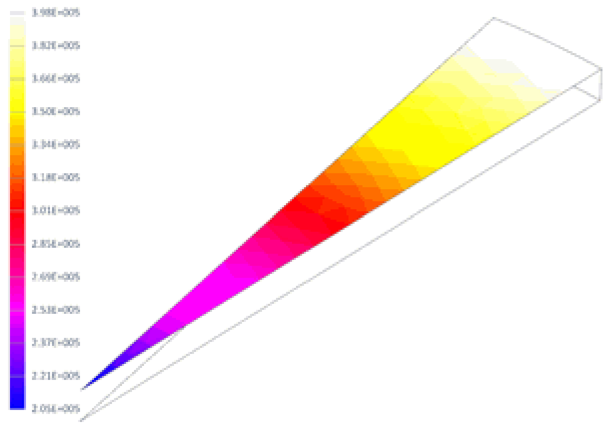
Swirl velocity results

What is it?

You can now recover swirl velocity results on 2D shell elements or on 1D axisymmetric elements. You request nodal, elemental, or nodal and elemental types of results from the **Swirl Velocity Results Location** list.

After you solve the model, depending on your request, either or both of the following result types appear under the solution node in the **Post Processing Navigator**:

- **Swirl Velocity on Walls - Nodal**
- **Swirl Velocity on Walls - Elemental**



The thermal solver computes swirl velocities when you define **Thermal Stream**, **Thermal Convective Zone**, or **Thermal Void** loads, or **Duct Flow Boundary Conditions** simulation objects of type **One-Sided Total Temperature Effects** or **Two-Sided Total Temperature Effects**.

Why should I use it?

These result sets help you analyze the behavior of your thermal model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Results Options** node→**Thermal** group→ **Swirl Velocities**

Results Location options

What is it?

The thermal solver can output the results at elements, at nodes, or at elements and nodes for additional results sets. In the previous release, the thermal solver output only elemental results for these results sets.

When you request one of the following results sets, you can now specify the location of the results.

On the **Thermal** tab:

- Total temperature
- Free and forced convection coefficient

On the **1D Flow** tab:

- Velocity
- Swirl velocity
- Pressure
- Density
- Mass flow
- Reynolds number

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution** 
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Results Options** node→**Thermal** group or **1D Flow** group

Heat transfer coefficient magnitude for edge interface resistance

What is it?

You can now specify the heat transfer coefficient for an **Interface Resistance** simulation object of type **Edge Interface**.

The resistance is calculated as:

1/(heat transfer coefficient x area)

At an edge interface, the area is calculated as follows:

area = (the length of the edge) x (the minimum shell thickness)

Why should I use it?

This improvement lets you define the resistance magnitude at an edge interface by specifying the heat transfer coefficient, which is a value that is often available in handbooks.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Command Finder	**Interface Resistance** 
Simulation Navigator	Right-click the **Simulation Objects** node→**New Simulation Object**→**Interface Resistance**
Location in dialog box	**Type** list→**Edge Interface**→**Magnitude** group→**Type** list→**Heat Transfer Coefficient**

Multiple Fluid material

What is it?

The new **Multiple Fluid** material type lets you define a primary fluid material and up to five additional fluid materials.

The thermal solver uses the specified primary fluid material properties during computation. The additional fluid material number can be passed as an argument to some CAE functions, such as fluid plug-ins, and also to user-defined functions. See User-defined functions for expressions for more information on plug-in functions.

You define the **Multiple Fluid** material type in the following loads and modeling object:

- **Thermal Convecting Zone** load
- **Thermal Stream** load

- **Void Non-Geometric Element** modeling object

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Multiphysics or NX Thermal and Flow as the specified solver.

Defining the multiple fluid material

Command Finder	**Manage Materials**
Location in dialog box	**Type** list→**Multiple Fluid**→ **Create**

Specifying the multiple fluid material in loads

Command Finder	**Thermal Convecting Zone** or **Thermal Stream**
Simulation Navigator	Right-click the **Load** container node→**New Load**→**Thermal Convecting Zone** or **Thermal Stream**
Location in dialog box	**Fluid Materials** list

Specifying the multiple fluid material in the modeling object

Command Finder	**Modeling Objects**
Simulation Navigator	Right-click the **Modeling Objects** node→**Modeling Objects**
Location in dialog box	**Type** list→**Void Non-Geometric Element**→**Create**→**Environment** group→**Fluid Materials** list

PCB Component and Printed Circuit Board simulation object support

What is it?

In NX Space Systems Thermal solver environment, you can create:

- **Printed Circuit Board** and **PCB Component** simulation objects.

- A **PCB Stack** physical property along with the **PCB Layer** and **PCB Via** modeling objects.

Why should I use it?

These new features let you easily define the thermal resistor models for printed circuit boards (PCBs) and electronic components in spacecraft applications.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Simulation objects

Application	Advanced Simulation
Command Finder	**Printed Circuit Board** or **PCB Component**
Simulation Navigator	Right-click the **Simulation Objects** node→**New Simulation Object**→**Printed Circuit Board** or **PCB Component**

PCB Stack physical property

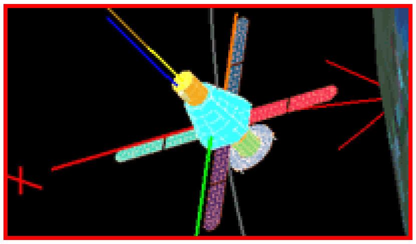
Application	Advanced Simulation
Command Finder	**Physical Properties**
Menu	**Insert**→**Physical Properties**
Location in dialog box	**Type**→**PCB Stack**→**Create**

Orbit Visualizer enhancements

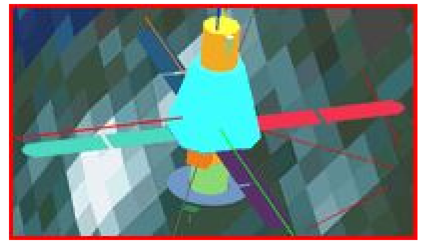
What is it?

The **Orbit Visualizer** now does the following:

- Keeps the spacecraft in view when you zoom, pan, or rotate the model.
- Lets you keep either the spacecraft or the planet in the center of the view with the new **Look at** view option.
- Displays elements of each mesh of the spacecraft model in a random and different color.
- Lets you make the element edges visible or invisible with the new **Element Edges** display option.



Visible element edges



Invisible element edges

Why should I use it?

You can distinguish the different components of the spacecraft model by their color.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Command Finder	**Orbital Heating**
Simulation Navigator	Right-click the **Simulation Objects** node→**New Simulation Object**→**Orbital Heating**
Location in dialog box	**View** group→**Display**

Automatic computation of view factors for flow surface elements

What is it?

The solver can now automatically create a radiation enclosure with all the flow surface elements in the model. To enable this functionality, select the **Create a radiation enclosure from all flow surface elements** check box.

The software splits the flow surface elements into two groups:

- The radiating top side region contains the flow surface elements convecting from the top.
- The radiating bottom side region contains the flow surface elements convecting from the bottom.

Note: The top side of an element is in the direction of the element's normal.

NX uses the following:

- The hemicube method to create the radiation enclosure.
- A medium (128 x 128) view factor accuracy.
- The global radiation parameter element subdivision method for element subdivision.

Why should I use it?

This option makes the task of creating a radiation enclosure that includes all the flow surface elements in your model more efficient. You do not have to create a radiation simulation object and define all the flow surface elements.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC

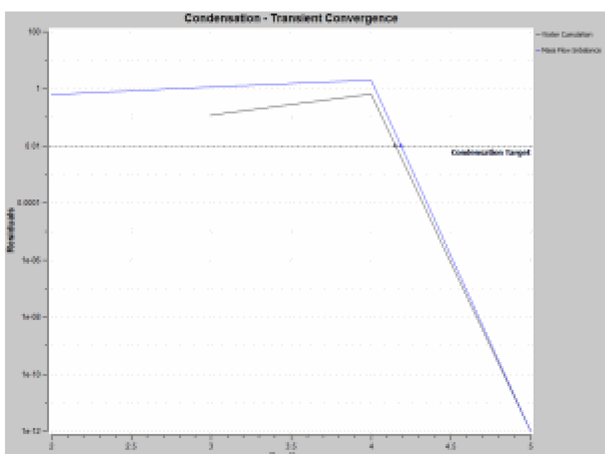
Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click a solution node→**Edit Solver Parameters**
Location in dialog box	**Radiation Parameters** node→**View Factor Calculation** group→**Create a radiation enclosure from all flow surface elements** ☑

Convergence graph enhancements

What is it?

The **Convergence** window now plots residual convergence graphs for the condensation variables as well as for the thermal, flow, and coupled variables. The condensation convergence variables are **Water Cumulation** and **Mass Flow Imbalance**.



In previous releases, the **Convergence** window plotted residual information only for the thermal, flow, and coupled variables.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type

NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	An open simulation file
Command Finder	**Solve**
Simulation Navigator	Right-click the **Solution** node→**Solve**
Location in dialog box	**Solution Monitor** dialog box→**Graph**→**Convergence**

Ignoring radiative environment in radiation modeling

What is it?

When you define a radiative ambient and set a radiative environment temperature, and clear the **Include Radiative Environment** check box in the **Radiation** dialog box, the solver now ignores any radiative exchange between the enclosure parts and the environment.

In the previous release, the solver automatically modeled radiative exchange between any part of the enclosure and the environment even if you cleared this check box.

Why should I use it?

Although you may have modeled the radiative environment in the enclosure, you can now exclude the environment in radiation modeling.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow
NX Multiphysics	Thermal	Thermal
	Coupled	Thermal-Structural (Multi-Step Nonlinear)

Where do I find it?

Application	Advanced Simulation
Command Finder	**Radiation**
Simulation Navigator	Right-click **Simulation Objects**→**New Simulation Object**→**Radiation**
Location in dialog box	**Parameters** group→**Include Radiative Environment**

Flow capabilities

Parallel flow solver enhancements

What is it?

The parallel flow solver now supports the following:

- Homogeneous gas mixture and tracer fluids
- High speed flows (fully coupled pressure-velocity parallel flow solver scheme)
- Shear stress transport (SST) and K-omega turbulence models
- Non-Newtonian fluids

You can also restart from a previous solution, and run the parallel flow solver using just one process.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Solution Details** node→**Flow Solver Selection** list→**Parallel Solver**

Two-phase, immiscible fluid simulation enhancements

What is it?

You can now do the following:

- Solve two-phase, immiscible fluid simulations in coupled thermal-flow analyses.
- Simulate fluid flow problems for a mixture of any two immiscible fluid constituents.
- Simulate open volume enclosures.
- Specify the surface tension.
- Specify the tracked fluid area fraction of the mixture at ambient condition.

Immiscible fluid mixture

In previous versions, the fluid constituents were limited to a liquid and a gas. Now, you can also specify two liquids or two gases, as long as the two fluids are immiscible.

In the **Immiscible Fluid Mixture** modeling object, you define one fluid as the tracked fluid and the other as the complementary fluid.

You also specify the surface tension between the tracked fluid and the complementary fluid.

When you post process the results, a volume fraction value close to one indicates that the mixture is mostly composed of the tracked fluid whereas a value close to zero indicates that the mixture is mostly composed of the complementary fluid.

Open volume enclosures

To simulate an open volume enclosure, you specify the tracked fluid area fraction of the mixture that enters the enclosure in the following ways:

- Specify it locally, for each inflow boundary condition, when you specify the **Inlet Flow** type, the **Opening** type, or the **Static Pressure** type of **Flow Boundary Condition** simulation object.
- Specify it globally, for all inflow boundary conditions, in the **Ambient Conditions** node of the **Solution** dialog box.

The mixture leaves the enclosure on faces where you specify the **Outlet Flow** type, the **Opening** type, the **Convective Outflow** type, or the **Static Pressure** type of **Flow Boundary Condition** simulation object. The flow solver computes the tracked fluid area fraction for the mixture that leaves the enclosure.

In previous versions, the two-phase, immiscible fluid mixture was restricted to a closed container such that the volume of the mixture was constant.

Additional text changes

The following option names have changed to support the enhanced functionality.

Pre-NX 10 name	NX 10 name	Command name and location in dialog box
Heterogeneous Gas-Liquid Mixture modeling object	**Immiscible Fluid Mixture** modeling object	**Modeling Objects→Type** list
Initial Gas-Liquid Mixture - 3D Flow type	**Initial Immiscible Fluid Mixture - 3D Flow** type	**Initial Conditions** constraint→**Type** list
Humidity and Mixtures group	**Humidity, Tracer Fluids, and Mixtures** group	**Solution→Ambient Conditions** node
Specify Fraction option	**Homogeneous Gas Mixture** option	**Solution→Ambient Conditions** node→**Humidity, Tracer Fluids, and Mixtures** group→**Mixture** list

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

These functionalities are available in Advanced Simulation when you use the parallel flow solver.

Specifying a two-phase, immiscible fluid mixture

Command Finder	**Modeling Objects**
Location in dialog box	**Type** list→**Immiscible Fluid Mixture→Create**

Setting the tracked fluid area fraction of the mixture entering the fluid enclosure

Command Finder	**Flow Boundary Condition**
Simulation Navigator	Right-click the **Simulation Object** container node→**New Simulation Object→Flow Boundary Condition**
Location in dialog box	**Type→Inlet Flow**, **Opening**, or **Static Pressure→Mixture** group

Setting the tracked fluid area fraction of the mixture at ambient condition

Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Ambient Conditions** node→**Humidity, Tracer Fluids, and Mixtures** group→**Mixture** list→**Immiscible Fluid Mixture**

Particle tracking enhancements

What is it?

You can now do the following:

- Solve only a particle tracking simulation.
- Specify particle impact types.
- Request particle impact and pass-through reports.
- Set new particle tracking controls.

Particle tracking simulation

You can solve steady state particle tracking simulation using the flow field from the solved steady state solution. If you have a solved flow analysis, you can do the following:

- Add **Particle Injection** simulation objects.
- Modify **Flow Surface** and **Screen** simulation objects.
- Modify particle tracking options in the **3D Flow** node of the **Solution** dialog box and in the **3D Flow Solver** node of the **Solver Parameters** dialog box.

After you modify your solution, select the **Solve Steady State Particle Tracking Simulation Only** check box and solve the solution again. The flow solver solves only the particle tracking equations.

Particle impact types

The flow solver can simulate particles that stick to the walls or that rebound. You can set the particle impact type in the following ways:

- Set it individually for each flow surface in the corresponding **Flow Surface** dialog box. You can define particle tracking options only in the **Embedded Flow Surface** and **Boundary Flow Surface** types.
- Set it globally for all flow surfaces in the **Solution** dialog box.

When you specify the rebound particle impact type, you must define the coefficient of restitution. The coefficient of restitution indicates how much energy the particle loses when it hits the wall. The specified value must be greater than or equal to zero and less than or equal to one.

Particle impact and pass-through reports

You can request a particle impact report in the **Flow Surface** simulation object and a pass-through report in the **Screen** simulation object. Only the **Embedded Flow Surface** and **Boundary Flow Surface** types have this option.

When you request the reports, the flow solver generates the *ParticleAndImpactTrackingSummary.dat* file which contains two tables:

- The **Particle Tracking Summary** table lists the number of injected and ejected particles in total for your model and per boundary condition.
- The **Particle Surface Impact and Screen Pass-Through Tracking Report** table lists the number of particles that impact walls in your model per injection boundary condition and the number of particles that pass through screens per injection boundary condition.

The file is generated in the simulation's run directory.

Optionally, you can output particle impact and pass-through results to the ParaView format. See ParaView output option.

Particle tracking controls

New options are available in the flow solver parameters for particle tracking simulation.

Steady State Simulation Time list
: Specifies the particle tracking simulation time. You can match the injection duration or specify an alternate total time for the particle tracking simulation. The value that you specify must be greater than the injection duration.

Neglect Stochastic Drag Terms option
: Neglects the Brownian and turbulent diffusivity terms in the particle tracking equations. Usually, you select this check box in conjunction with the **Use Cunningham Correction Factor** check box.

Use Cunningham Correction Factor option
: Applies the computed Cunningham correction factor to the particle traction force estimate. The Cunningham correction factor represents the reduction in the force exerted upon the particle by the flow due to the breakdown of the no-slip condition on the particle surface. The no-slip condition is invalid when the mean free path of the fluid molecules is of the same order as the particle length scale. This option is valid only for ideal gases. The flow solver computes the correction factor using the following constants that you specify:
  - Sutherland constant for the viscosity of the gas
  - Slip correction reference temperature, pressure, and dynamic viscosity of the gas

In addition, the following table lists the name changes in the **3D Flow Solver** node of the **Solver Parameters** dialog box.

Pre-NX 10 name	NX 10 name
Steady State Particle Controls group	**Particle Tracking Controls** group
Injection Duration box	**Steady State Injection Duration** box
Output Option list	**Steady State Output Option** list

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

These functionalities are available in Advanced Simulation when you use the serial flow solver.

Simulation objects

Command Finder	**Flow Surface** or **Screen**
Simulation Navigator	Right-click the **Simulation Object** container node→**New Simulation Object**→**Flow Surface** or **Screen**
Location in dialog box	**Particle Tracking** group

Solution options

Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Restart** node→ **Perform Restart**→**Options** group→ **Solve Steady State Particle Tracking Simulation Only** **3D Flow** node→**Flow Surface Parameters** group

Particle tracking controls

Simulation Navigator	Right-click a solution node→**Edit Solver Parameters**
Location in dialog box	**3D Flow Solver** node→**Particle Tracking Controls** group

ParaView output option

What is it?

You can now output results to the ParaView format. When you request output in ParaView format, NX can create the following ParaView files. The number of ParaView files created depends on the solution.

- The *flow_solid.vtu* file contains the nodal solution field for a steady state run.
- The *flow_solid_M.vtu* files contain the nodal solution field at the output time *M* for a transient run. NX creates one file for each output time.

- The *partdata_N.vtp* files contain the coordinates of the particle positions at the particle tracking output time *N*. NX creates one file for each output time. These files are generated for a particle tracking simulation.

- The *paraview_SiteName_N.vtp* files contain the position coordinates of all particles injected from the site *SiteName* at the particle tracking output time *N*. NX creates one file for each particle injection site and output time. These files are generated for a particle tracking simulation with requested particle impact or pass-through reports.

- The *paraview_SiteName_TrackingName_N.vtp* files contain the position coordinates of particles injected at site *SiteName* which contact the tracking site *TrackingName* at the particle tracking output time *N*. NX creates one file for each particle injection site that has a particle in contact with the tracking site, and for each output time. These files are generated for a particle tracking simulation with requested particle impact or pass-through reports.

The files are generated in the simulation's run directory.

SiteName is the name of the **Particle Injection** simulation object. *TrackingName* is the name of the **Flow Surface** or **Screen** simulation object from which you request the reports. See Particle tracking enhancements.

Why should I use it?

The ParaView format allows you to import your results into the ParaView third party software for post processing of injected particles.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Results Options** node→**Optional Output Format** group→ **ParaView**

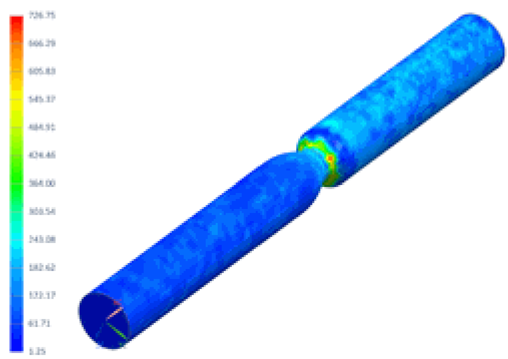
Non-Newtonian model quantity results

What is it?

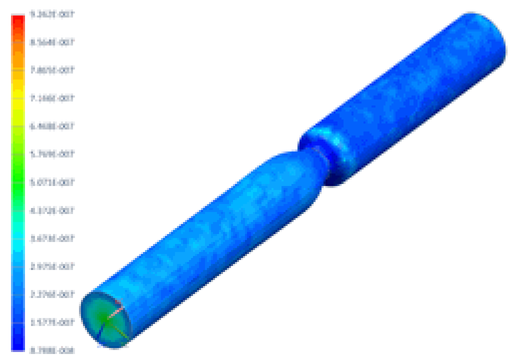
You can now recover shear rate and dynamic viscosity results on 3D elements when you use non-Newtonian fluids in your model.

After you solve the model, the following results types can appear under the Solution node in the **Post Processing Navigator**:

- **Shear Rate - Elemental-Nodal**
- **Dynamic Viscosity - Elemental-Nodal**



Shear rate



Dynamic viscosity

Why should I use it?

The display of shear rate and dynamic viscosity results can help you analyze models with non-Newtonian fluids.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must define a non-Newtonian fluid and use the parallel flow solver.
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Results Options** node→**3D Flow** group→ **Non-Newtonian Model Quantities**

Restart option for the parallel flow solver

What is it?

You can now restart a flow or coupled thermal-flow solution that you previously solved using the parallel flow solver. You can restart only those solutions for which you created backup files. You request the creation of backup files from the **Create Backup Files** list. The following options are available.

For Paused or Completed Solution	Creates backup files for both steady state and transient solutions when you pause the solution or when the solution finishes.
At Specified Number of Iterations	Creates a backup file for a steady state solution at every N number of iterations. N is the value that you specify in the **Number of Iterations** box.
At Results Sampling Times	Creates a backup file for a transient solution at each results sampling time. You set the sampling times on the **Transient Setup** node in the **Results Sampling** group.
Never	Does not create backup files. You cannot restart solutions if you use this option.

The parallel flow solver backup files are *flow.restart* and *flow.struct*. They contain all the necessary information for restarting the simulation from the point at which the backup files were updated. The information they contain includes, but is not limited to, flow field variables, mass flow rates, and gradient vectors. NX creates the files in same run directory that contains all the other flow solution files.

You cannot modify the original mesh of the model for which you want to restart the solution. You can however make the following changes to your solution:

- Add or remove boundary condition simulation objects.
- Modify the boundary condition values.
- Modify the time step size, relaxation factors, time integration accuracy, and advanced parameters.
- Change the flow type, for example, from incompressible to high speed flow.
- Change the turbulence model.
- Change the number of processes for a parallel run.
- Add meshed geometry or flow enclosures as long as the original mesh is not altered in anyway, for example, by reordering nodes or elements.

  Note

  You cannot restart solutions using the parallel flow solver that were solved using the parallel flow solver from previous NX releases or using the serial flow solver.

Why should I use it?

Perform a restart when you want to:

- Pause, inspect, and if necessary modify your solution before continuing the solve.
- Restart your solution with incremental changes in the boundary conditions.
- Restart a simulation that diverged or crashed previously.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must use the parallel flow solver.
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	(To create backup files) **Restart** node→**Parallel Flow Solver Restart Backup File Options** group→**Create Backup Files** (To restart from backup files) **Restart** node→ **Perform Restart**

CGNS results file options (10.0.1)

What is it?

You can now request transient flow results in the CGNS format to be written in one of the following ways:

- One CGNS results file: The results file contains results from all transient output times.
- Multiple CGNS results files: Each results file contains results from one transient output time.

In previous releases, NX always wrote a single CGNS results file containing results from all transient output times.

Why should I use it?

You can now obtain the results from each transient output time in its own CGNS results file. You do this by requesting multiple CGNS files. You can continue to request a single CGNS file that contains all the results from all transient output times.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Results Options** page→**Optional Output Format** group→**CGNS** →**Transient Results** list

NX Laminate Composites

Samcef solver support in NX Laminate Composites

What is it?

You can access most NX Laminate Composites commands and functionality in the Samcef solver environment.

All laminate commands are supported except the following:

- **Solid Laminate Physical Property**
- **Ply Materials**

Ply IDs in all laminate physical properties and global layups must have unique values. In other solver environments, ply IDs in different laminate physical properties can have the same values.

In the Samcef environment, you can create cohesive layers between extruded plies to model delamination. See Cohesive layers for more information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file as the work part and displayed part Samcef as the specified solver

Cohesive layers

What is it?

You specify cohesive layers between plies of your global layup. For each cohesive layer, you need to specify a damage interface material.

Global Ply ID	Composition	Thickness	Angle	Status	Description	Solid Property
3	PPG-5H-3K	1	0	Up-to-date		
c5	Cohesive1	0	0	Cohesive Layer	Cohesive Layer	
2	PPG-5H-3K	1	0	Up-to-date		
c4	**Cohesive1**	**0**	**0**	**Cohesive Layer**	**Cohesive Layer**	
1	PPG-5H-3K	1	0	Up-to-date		

When you inflate the laminate, NX creates meshes of a layer of cohesive elements between layers of ply meshes. Each cohesive mesh is stored in a separate mesh collector that points to a **Cohesive Property** physical property. The physical property stores the material that you specified for the cohesive layer.

- 3D Collectors
  - PART (laminate shell)1–Extrusion
    - PART (laminate shell)1–Ply1
    - PART (laminate shell)1–Ply2
    - **PART (laminate shell)1–Ply3**
  - PART (laminate shell)1–Junction4
    - PART (laminate shell)1–Junction4
  - PART (laminate shell)1–Junction5
    - PART (laminate shell)1–Junction5

Why should I use it?

Use the cohesive layers to model delamination in the Samcef solver environment.

Where do I find it?

Creating cohesive layers

Application	Advanced Simulation
Prerequisite	You must work in the FEM within the Samcef solver environment.
Command Finder	**Global Layup**
Location in dialog box	**Layup Definition** group→**Create New Cohesive Layer**

Inflating laminates

Application	Advanced Simulation
Prerequisite	You must work in the FEM within the Samcef solver environment.
Command Finder	**Extrude Laminate** or **Fill Laminate**

Import Fibersim Zones

Use the **Import Fibersim Zones** command to import the zones from a Fibersim HDF5 file onto selected polygon faces or 2D elements. The zones must point to a laminate that is associated to the same part geometry that is meshed in NX.

When you import Fibersim zones, NX creates:

- One mesh and one mesh collector for each zone.
- One laminate physical property for each Fibersim laminate specification.
- One material table for each Fibersim material specification, if you request it.
- Curves that represent the zone boundaries, if you request this.

Each mesh collector points to one of the created laminate physical properties. NX moves elements from the existing NX mesh into the newly created meshes. The existing NX mesh is empty after the import.

In the **Import Zones** dialog box, you can specify if you want to overwrite existing materials, laminate physical properties, or mesh collectors.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Command Finder	**Import Fibersim Zones**

Import layups from Fibersim

What is it?

You can now import layups directly from an NX part containing Fibersim data, without passing through an HDF5 file. You specify the Fibersim laminate that you want to import from the **Fibersim Objects** list in the **Import Layup** dialog box.

The import works only when the following conditions are met:

- Fibersim 14 or above must be installed.
- The PATH environment variable includes the path to the NX folder in the Fibersim installation folder. It is the same path as defined in the UGII_VENDOR_DIR environment variable.
- The **Use Fibersim API Import** customer default is selected.

- The NX part containing Fibersim data is loaded. The part file must contain an NX Laminate Composites ply export object. This object defines the laminates and the components to be exported. You define it in Fibersim using **File→Export→Analysis – Ply→NX Laminate Composites**.

  Tip

  To find a customer default, choose **File** tab→**Utilities→Customer Defaults**, and click **Find Default** .

Where do I find it?

Application	Advanced Simulation
Command Finder	**Import Global Layup** 
Location in dialog box	**File Type** list→**Current Fibersim Part**

Editing multiple laminate physical properties

You can edit multiple laminate physical properties or solid laminate physical properties at the same time.

For the selected physical property tables, you can modify:

- Solver properties
- Laminate properties, including:
  - Stacking recipe
  - Zone angle tolerance
  - Reference plane location for 2D laminate properties
  - Reference temperature
  - Ply and interlaminar failure theories including user-defined failure theories, interlaminar allowables, shear stress for bonding, and normal stress for bonding
- Ply Layup. You can reverse either the plies and their ply IDs or just the plies keeping the original ply ID sequence.
- Ply IDs. You can relabel the ply IDs using one of the following:
  - A start label and an increment between ply IDs. NX relabels the existing ply IDs starting with the given start label and increments each subsequent ply ID by the increment value.
  - An offset. NX relabels the existing ply IDs by offsetting their old value by the specified offset.

The following table lists the significance of icons that can be displayed in front of each property in the **Multiple Edit Laminate Properties** dialog box.

Icon	Hover text	Description
	No Change	Indicates that the value currently defined for this property is the same for all the selected laminate physical properties.
	No Change	Indicates that the value currently defined for this property is different for two or more of the selected laminate physical properties.
	Apply Change	Indicates that the change you made for this property will be applied when you click **OK**.
	<Laminate physical property name>:<property name>	Indicates that the value of this property for all physical properties will be the same as for the named laminate physical property, after you click **OK**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Command Finder	**Physical Properties**
Location in dialog box	Select multiple laminate physical properties or multiple solid laminate physical properties→**Edit**

Expressions support for ply thickness, ply angle, and layup offset

What is it?

When you use an expression to define the value of the ply thickness, ply angle, or layup offset, NX now updates this value if you change the expression.

Note

NX Laminate Composites does not support expressions that point to fields.

In previous releases, if you used expressions to define these values, the constant value computed the first time was always used even if you subsequently changed the value of the expression.

Where do I find it?

Ply thickness and ply angle

Application	Advanced Simulation
Prerequisite	A FEM as the work part

Command Finder	**Laminate Physical Property** or **Global Layup** or **Solid Laminate Physical Property**
Location in dialog box	(**Laminate Modeler** or **Solid Laminate Modeler** dialog box) **Ply Layup** tab→**Thickness** or **Angle**→click and choose **Formula** (**Layup Modeler** dialog box) **Layup Definition** group→**Thickness** or **Angle**→click and choose **Formula**

Layup offset

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Simulation Navigator	Right-click the **Layup Offset** node→**Create User Defined Layup Offset Rule**
Location in dialog box	**Bottom Fiber Distance**→click and choose **Formula**

Relabel global ply IDs

What is it?

Use the **Relabel Ply IDs** command to change the global ply IDs for all global plies of selected layups.

You can relabel ply IDs by specifying one of the following:

- A start label and an increment between ply IDs. NX relabels the existing ply IDs starting with the given start label and increments each subsequent ply ID by the increment value.

  Note

  Symmetric and repeated plies need to have consecutive ply IDs. In this case, the increment value should be one.

- An offset. NX relabels the existing ply IDs by offsetting their old value by the specified offset.

Why should I use it?

You can now make post-processing results easier to understand by relabeling the global ply IDs in your FEM structure.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Simulation Navigator	Right-click the **Layups** node or one or more global layup nodes→**Relabel Ply IDs**

Inflated laminates in LS-DYNA (10.0.1)

What is it?

The **Extrude Laminate** and **Fill Laminate** commands are now supported in the LS-DYNA environment.

The extruded 3D mesh consists of ELEMENT_TSHELL wedge or hexahedral elements and the ply drop-offs consist of ELEMENT_SOLID tetrahedral, wedge, or hexahedral elements. Because LS_DYNA does not support pyramids, pyramids are split into tetrahedral elements.

For both commands, NX creates a 3D mesh collector that has a **Solid Laminate** physical property assigned to it. NX automatically sets the **Stacking Recipe** in the **Solid Laminate Modeler** dialog box to **Inherited from Layup**.

All the plies of the extruded solid laminate have the same ply failure theory and inter-laminar failure theory that you selected in the **Laminate Modeler** dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM within the LS-DYNA solver environment.
Command Finder	**Extrude Laminate** or **Fill Laminate**

New laminate material options for LS-DYNA

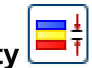
What is it?

This release includes new options in the **Laminate Modeler** and the **Solid Laminate Modeler** dialog boxes that control how NX exports the laminate material to LS-DYNA.

- Use the new **Shell/Solid Formulation** option to specify the shell or solid formulation output format when you export the laminate to the LS-DYNA solver.
  - Select **Layered** to export the laminate as layered properties.
  - Select **Homogenous** to export the laminate as unlayered, homogeneous properties with equivalent orthotropic materials. With this option, NX computes the equivalent properties of a single ply. Exporting 3D laminates as homogeneous properties rather than layered properties may shorten the solution time. The homogeneous approach is suitable for balanced and symmetric laminates
- Use the new **NX Orthotropic Materials Mapping** option to map the NX orthotropic material to an appropriate LS-DYNA material type, such as `*MAT_ORTHOTROPIC_ELASTIC (MAT_002)` or `*MAT_COMPOSITE_LAYUP (MAT_116)`.

Where do I find it?

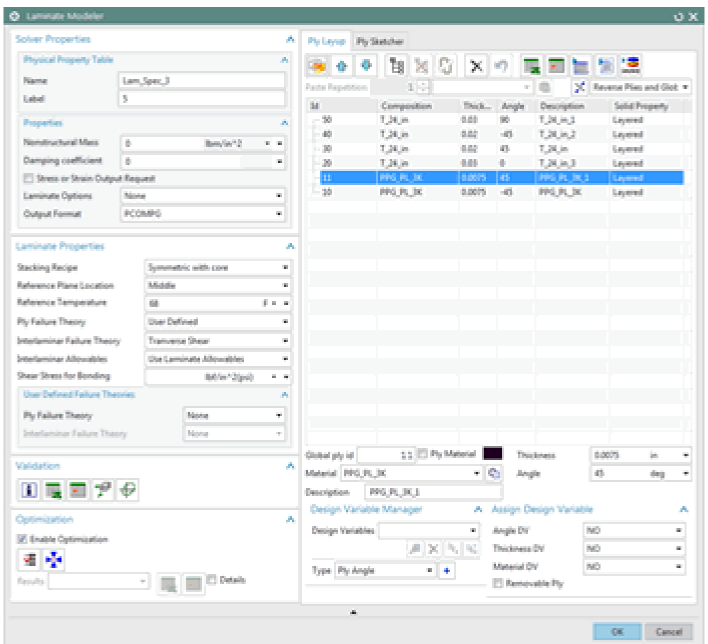
Application	Advanced Simulation
Prerequisite	A FEM file as the displayed part and work part LS-DYNA as the specified solver

Command Finder	**Laminate Physical Property**  or **Solid Laminate Physical Property** 
Location in dialog box	**Solver Properties** group

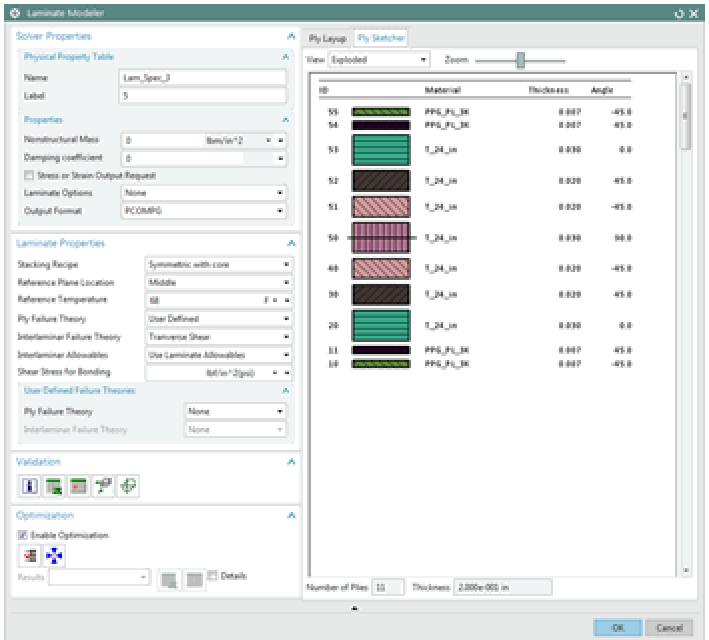
Laminate Modeler dialog box enhancement

What is it?

The groups in the **Laminate Modeler** dialog box are rearranged to reduce clutter, allow for more visible plies, and for ease of use. The **Ply Layup** and **Ply Sketcher** groups are now tabs in the dialog box. All other groups are located on the left side of the dialog box.



Ply Layup tab



Ply Sketcher tab

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Command Finder	**Laminate Physical Property** or **Solid Laminate Physical Property**

Adjusting the font in Ply Sketcher

What is it?

You can now set the font type and size for your layup information on the **Ply Sketcher** tab in the following dialog boxes:

- **Laminate Modeler** dialog box
- **Solid Laminate Modeler** dialog box
- **View Laminate** dialog box
- **View Laminate — Zone #** dialog box

You set the font type and size in the **Visualization Preferences** dialog box. In previous releases, NX set the font automatically and you could not modify it.

Where do I find it?

Visualization Preferences dialog box

Application	All
Command Finder	**Visualization Preferences**
Location in dialog box	**Color/Font** tab

Laminate Modeler dialog box and **Solid Laminate Modeler** dialog box

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Command Finder	**Laminate Physical Property** or **Solid Laminate Physical Property**

View Laminate dialog box and **View Laminate — Zone #** dialog box

Application	Advanced Simulation
Command Finder	**View Laminate**
Simulation Navigator	(FEM) Right-click the **Zones** or **Zone #** node→**View Laminate**

Element-nodal results in laminate post reporting

What is it?

When the results from the structural solver are element-nodal results, you can now output the laminate post reporting results as element-nodal results.

In previous versions, NX averaged the element-nodal results for each element before it used them to compute the laminate post reporting results.

Why should I use it?

You can display the results from the laminate post reporting in the same way as you display the structural results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Element-nodal structural results
Command Finder	**Spreadsheet Report** or **Graphical Report**
Simulation Navigator	Right-click the **Spreadsheet Report** node or the **Graphical Report** node→**Edit**
Location in dialog box	**Options** tab→**Input Selection** group→**Element Nodal Results**

Multithreaded draping process

What is it?

To improve the draping performance for the woven and unidirectional solvers, you can enable multithreading which uses shared-memory parallelization.

Set the number of execution threads using the **Number of Execution Threads for Multithreading** customer default. Multithreading is enabled when the number of execution threads is greater than one.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Command Finder	**Update Global Layups and Zones**
Simulation Navigator	Right-click the **Layups** node or one or more layup nodes or one or more ply nodes→**Update**

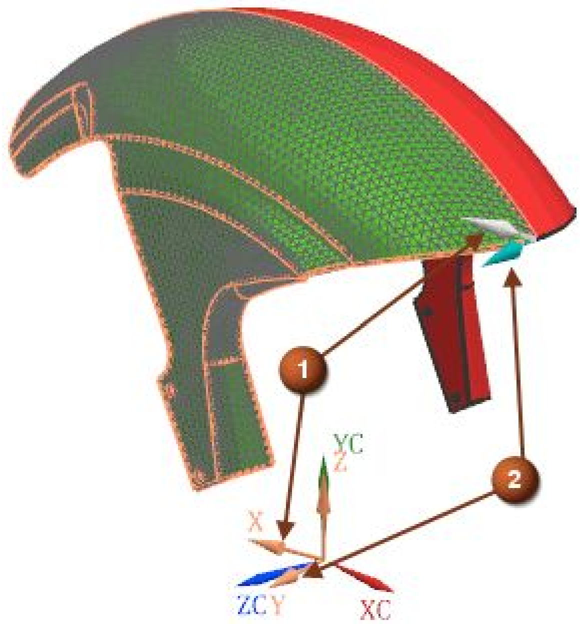
Specifying ply directions using a coordinate system

What is it?

When you align fibers for draping, in the **Draping Data** dialog box, you can now specify a coordinate system instead of a vector.

For the unidirectional draping solver, the primary fiber direction is aligned with the X-axis (1).

For the woven draping solver, the primary fiber direction is aligned with the X-axis (1) and the secondary fiber direction is aligned with the Y-axis (2).



Why should I use it?

You have more flexibility when you define the ply directions for draping. In particular, you can select a coordinate system that corresponds to an imported Fibersim rosette.

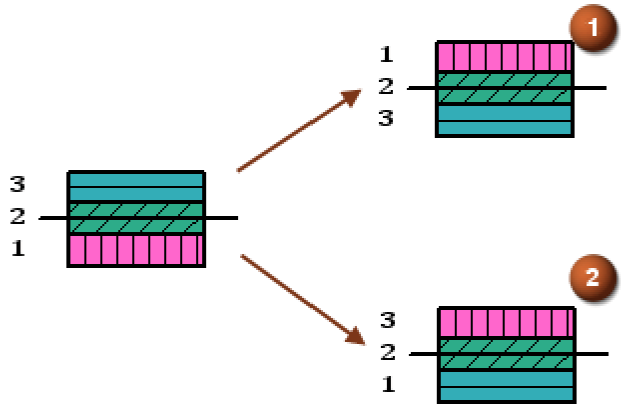
Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Simulation Navigator	Right-click a global ply node→**Edit**
Location in dialog box	**Solver** list→**Woven** or **Unidirectional**→**Direction Option** list→**CSYS**

Reversing the stacking sequence

What is it?

You can now easily reverse the stacking sequence of a layup with the introduction of the new Reverse Stacking Sequence  command. You can reverse the plies and their global ply IDs (1), or reverse the plies but keep the original ply ID sequence (2).



In previous releases, reversing the ply sequence and optionally the IDs was more of a manual operation because this command did not exist.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Command Finder	**Laminate Physical Property** or **Solid Laminate Physical Property** or **Global Layup**
Location in dialog box	(**Laminate Modeler** or **Solid Laminate Modeler** dialog box) **Ply Layup** tab→Reverse Stacking Sequence (**Layup Modeler** dialog box) **Layup Definition** group→Reverse Stacking Sequence

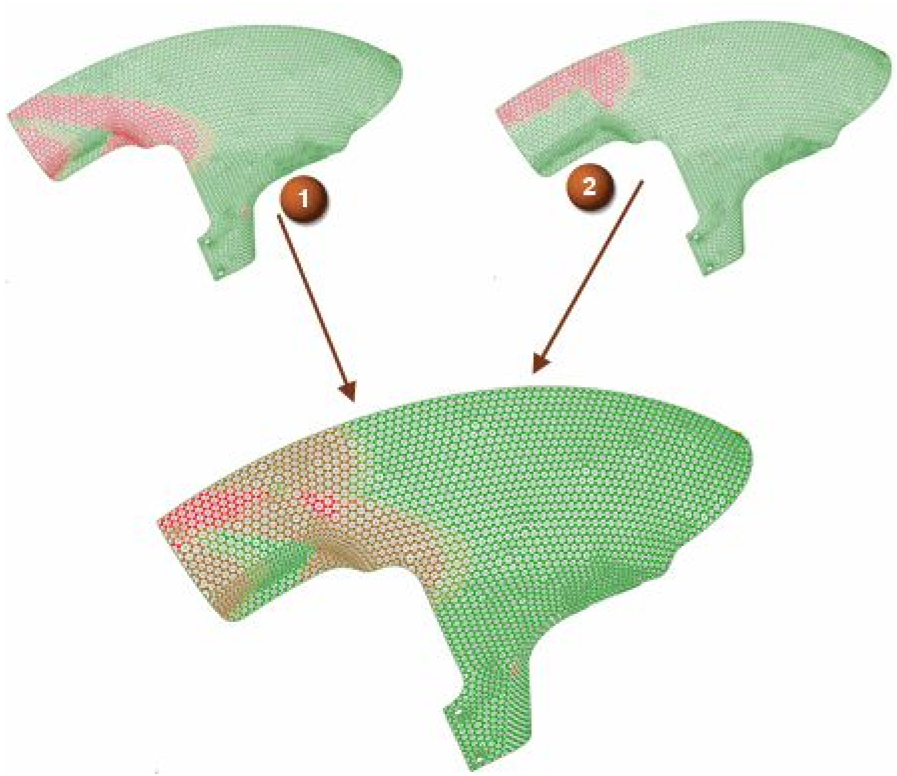
Viewing fiber orientations on multiple plies

What is it?

You can now display the fiber orientations for multiple plies at the same time. NX superposes the fiber orientations of each ply.

Example

The fiber orientations of ply (1) and ply (2) are superposed when you request the fiber orientations for both plies at the same time.



In previous versions, you could view fiber orientations only for one ply at a time.

Why should I use it?

You can compare the orientations of one or more plies.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Simulation Navigator	Right-click multiple global ply nodes→**View Fiber Orientations**

Grouping of unmapped Fibersim elements

What is it?

When you import plies of a Fibersim ply-based laminate onto selected polygon faces or 2D elements, NX creates a group of unmapped elements for those elements that have no mapped ply data.

- Groups
  - 1 - Unmapped Elements

Where do I find it?

Application	Advanced Simulation
Command Finder	**Import Layup**

Ply description in the spreadsheet

What is it?

When you export or import a ply layup to or from a spreadsheet application, the ply description is also exported or imported.

In previous releases, for each ply, only the material name, ply thickness, and ply angle were imported or exported.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM as the work part
Command Finder	**Laminate Physical Property** or **Global Layup** or **Solid Laminate Physical Property**
Location in dialog box	(**Laminate Modeler** or **Solid Laminate Modeler** dialog box) **Ply Layup** tab→**Export Layup to Spreadsheet** or **Import Layup from Spreadsheet** (**Layup Modeler** dialog box) **Layup Definition** group→**Export Layup to Spreadsheet** or **Import Layup from Spreadsheet**

Generating multiple graphical reports

What is it?

You can now generate results from multiple graphical reports at the same time.

In previous releases, you could generate results for only one graphical report at a time.

Why should I use it?

You save time when you want to generate results for multiple graphical reports.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must select multiple graphical nodes in the **Simulation Navigator**.
Simulation Navigator	Right-click multiple graphical report nodes→**Generate Result File**

More failure metric results for maximum stress and maximum strain criteria (3D)

What is it?

NX now provides all six failure index values for solid laminate plies that have a specified maximum stress or maximum strain criterion. The following table lists the equations used to compute these values.

Failure index value	Maximum stress criterion	Maximum strain criterion
F_1	σ_1/X_T if $\sigma_1>0$ σ_1/X_C if $\sigma_1<0$	$\varepsilon_1/X_{T\varepsilon}$ if $\varepsilon_1>0$ $\varepsilon_1/X_{C\varepsilon}$ if $\varepsilon_1<0$
F_2	σ_2/Y_T if $\sigma_2>0$ σ_2/Y_C if $\sigma_2<0$	$\varepsilon_2/Y_{T\varepsilon}$ if $\varepsilon_2>0$ $\varepsilon_2/Y_{C\varepsilon}$ if $\varepsilon_2<0$
F_3	σ_3/Z_T if $\sigma_3>0$ σ_3/Z_C if $\sigma_3<0$	$\varepsilon_3/Z_{T\varepsilon}$ if $\varepsilon_3>0$ $\varepsilon_3/Z_{C\varepsilon}$ if $\varepsilon_3<0$
F_4	$\lvert\tau_{12}/S_{12}\rvert$	$\lvert\gamma_{12}/S_{\varepsilon 12}\rvert$
F_5	$\lvert\tau_{13}/S_{13}\rvert$	$\lvert\gamma_{13}/S_{\varepsilon 13}\rvert$
F_6	$\lvert\tau_{23}/S_{23}\rvert$	$\lvert\gamma_{23}/S_{\varepsilon 23}\rvert$

For maximum stress criterion:

- X_T and X_C are the maximum allowable stresses in direction 1 in tension and compression.
- Y_T and Y_C are the maximum allowable stresses in direction 2 in tension and compression.
- Z_T and Z_C are the maximum allowable stresses in direction 3 in tension and compression.
- S_{12} is the maximum allowable in-plane shear stress.
- S_{13} and S_{23} are the maximum allowable 13 and 23 shear stresses.

For maximum strain criterion:

- $X_{T\varepsilon}$ and $X_{C\varepsilon}$ are the maximum allowable strains in direction 1 in tension and compression.
- $Y_{T\varepsilon}$ and $Y_{C\varepsilon}$ are the maximum allowable strains in direction 2 in tension and compression.
- $Z_{T\varepsilon}$ and $Z_{C\varepsilon}$ are the maximum allowable strains in direction 3 in tension and compression.
- $S_{\varepsilon 12}$ is the maximum allowable in-plane shear strain.
- $S_{\varepsilon 13}$ and $S_{\varepsilon 23}$ are the maximum allowable 13 and 23 shear strains.

In previous releases, for both criteria, NX Laminate Composites provided only one value of the failure index. That value was the maximum of the six failure indices.

Where do I find it?

Application	Advanced Simulation
Prerequisite	The ply failure theory must be set to **Max Stress** or **Max Strain** for solid laminates.

Command Finder	**Spreadsheet Report** or **Graphical Report**
Simulation Navigator	Right-click the **Spreadsheet Report** node or the **Graphical Report** node→**Edit**
Location in dialog box	**Options** tab→**Output Control** group→ **Failure Index** or **Strength Ratio** or **Margin of Safety**

Durability

Saving geometry in durability results files

What is it?

When you run a durability solution process, NX can now generate the solver results in BUN files, which contain the geometry. To enable this functionality, set the **Save Geometry in Results Files** customer default.

In previous versions, NX generated durability solver results only in BUD files.

BUN files are larger in size than BUD files.

Why should I use it?

NX Post Processing can use either BUD or BUN files to display durability result sets. If you use BUD files, you need to load the parent solution result file to display durability result sets.

Because BUN files also contain geometry you can display durability results in NX Post Processing without loading the parent solution result file.

Where do I find it?

Application	Advanced Simulation
Command Finder	**Customer Defaults**
Location in dialog box	**Simulation**→**NX ADVANCED DURABILITY**→**Solver Parameters** tab→**Save Geometry in Results Files** check box

Post-processing

General post-processing enhancements

What is it?

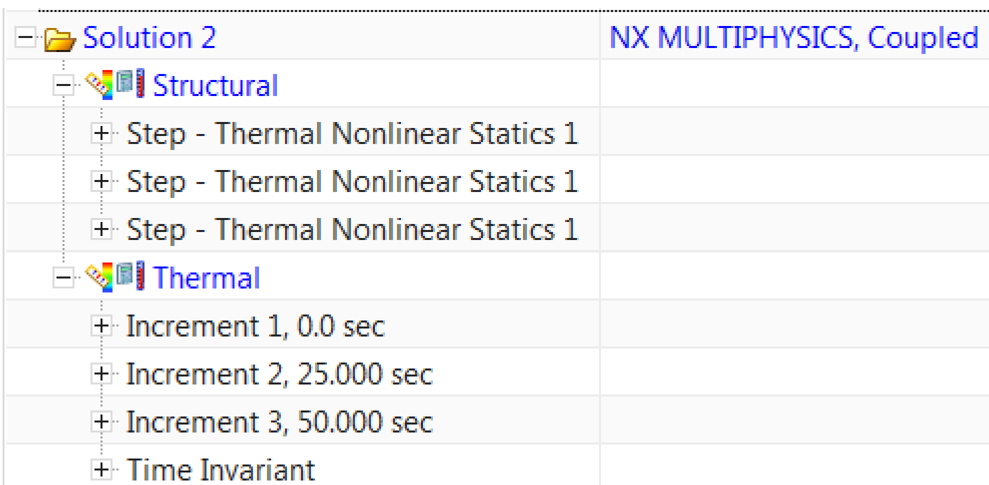
This release includes a number of general enhancements to NX Post-processing. These include:

- NX Multiphysics results support
- Global text formatting preferences for post views
- Support for Kelvin

- Graphing enhancements
- New averaging options
- New derived results for worst principal stress
- Samcef results support

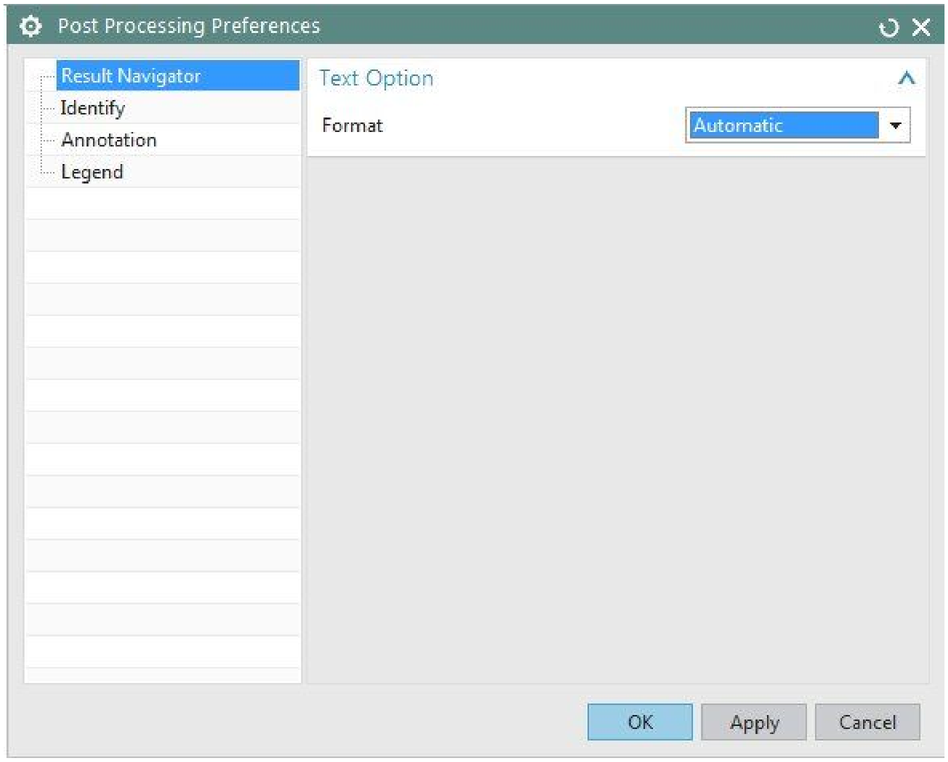
NX Multiphysics results support

In NX Multiphysics solutions, the structural and thermal results are listed together under the same solution in the **Post Processing Navigator**. You can right-click a result node and choose **Load** to load the specific results file, or right-click the **Solution** node and choose **Load All** to load all of the results for that solution.



Global text formatting for post views

You can now specify global text formatting preferences for all post views using the new **Post Processing Preferences** dialog box. These preferences override the **Legend Text** options for individual post views in the **Post View** dialog box.



Support for Kelvin

You can now display and manipulate results in Kelvin in NX Post-processing, when you have one or more results files loaded.

Graphing enhancements

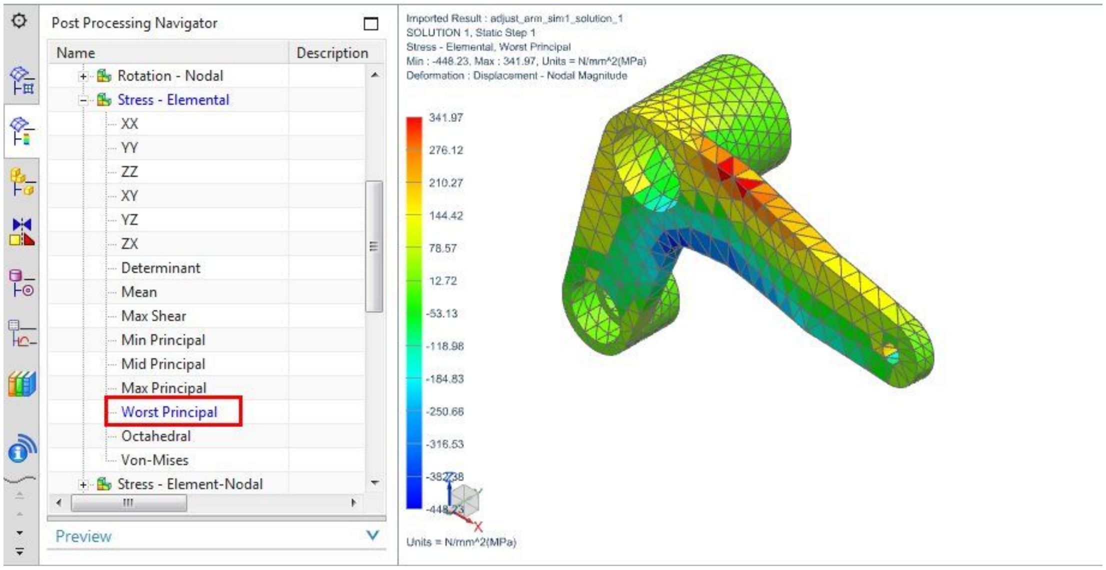
You can now select **Include Intersections** in the **Graph** dialog box to create a smooth graph of the changes in data values between points. This new option is available when the graph **Type** is set to **Path** and the **Define by Path** option is selected.

New averaging options

Two new averaging options are added to the **Multiple Reduction** dialog box: **Weighted Average** and **Integral**. When you select either of these options, you can define the type of weighted averaging (for example, **Area**, **Volume**, or **Length**) based on the selection type. For example, if you select **Faces** as your selection type, you can define your weighted average by **Area**. In addition, the **Average** option has been renamed to **Arithmetic Mean**.

New derived results for worst principal stress

For stress results at nodes on elements, NX Post-processing now derives the worst principal stress tensor component. The worst principal stress is the principal stress component with the largest absolute value.



Samcef results support

You can now display results from Samcef in NX Post-processing. You can:

- Import Samcef results into NX. The results are listed under the **Results** node in the **Simulation Navigator**.
- Create post-processing displays of the Samcef results in NX, such as contour plots and animations. You can also use the results manipulation capabilities with the Samcef results.

Where do I find it?

Post Processing Preferences dialog box

Application	Advanced Simulation
Command Finder	**Post Processing Preferences**
Menu	**Preferences→Post Processing**

Multiple Reduction dialog box

Application	Advanced Simulation
Command Finder	**Results Multiple Reduction** 
Menu	**Tools→Results→Multiple Reduction**

Generating results with result probes and variables

What is it?

This release provides a new way to display results with two new commands:

- **Result Probe**

- **Result Variables**

Result Probe

Use the **Result Probe** command to generate customized results by using scalar expressions.

The result probe is associated with a solution. You can define a result probe either before or after solving the model.

To use a results probe, the model must be solved. When you are ready to generate results, in the **Simulation Navigator**, right-click the results probe and select the type of output required, which can be contour plots, XY graphs, fields, or text. The results probe evaluates the results, and then generates the output.

Result probes are stored in the Simulation file.

Result variables

Use the **Result Variables** command to define variables that you can use as input to a result probe.

To define a result variable, you must specify the following:

- A result type such as displacement, temperature, etc.
- A location, which can be nodal, elemental, or element nodal.
- A result type component such as X,Y,Z, Von mises, etc.

You can optionally specify shell-specific and beam-specific options, averaging, complex options, and the output coordinate system.

A result variable can be used to define a formula for a result probe. A result probe is associated with a solution, and it specifies results, load cases, iterations, or geometry to be used to generate the results.

The result variable is not associated with any solution. It is saved in a Simulation file. You can use a result variable in multiple solutions and result probes.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and work part Results
Command Finder	**Result Probe** or **Result Variables**
Simulation Navigator	Right-click the solution node or the **Result Probes** node→**New Result Probe**.

Teamcenter Integration for Simulation

Additional file types supported as named reference

What is it?

There are new types of solver files that you can save to Teamcenter.

These solver types are supported as named references when you use Teamcenter Integration with Teamcenter 10.1.3, Teamcenter 11.1, or later releases. If you are using an older release of Teamcenter, your Teamcenter administrator can add these file types to the named references to support these solvers.

Solver	File Type
NX Nastran	Nastran results (.f56)
SAMCEF	SAMCEF workflow (.samx) Bank file (.dat) Database file (.sdb) Complementary database file (.adb) Results file (.res) Storage file (.u18) Descriptive file (.des) Post-processing file (.fac) Log file (.log) Diagnostic file (.dia)
Acoustics	Input deck and results for acoustics solvers (.sdb) Command file (.cmd)
Virtual Lab	Solver input file (.minp) Contact tessellation input (.off) Solver result file (.mres) Solver information file (.minf) Model representation (.mxml)
NX Multiphysics	Log file (.mplg) Input deck (.mpdat)

Assigning custom item types for CAE parts during import

What is it?

When you use the **Import Assembly into Teamcenter** command, if you have custom item types for CAE parts, you can now select the item types for the type of file you import. You select the item types from the following columns in the **Import Assembly into Teamcenter** dialog box:

- The **Item Type** column for the master part file.
- The **CAEGeometry Item Type** column for the idealized part file.
- The **CAEModel Item Type** column for the FEM file.

- The **CAEAnalysis Item Type** column for the simulation file.

The **Import** customer defaults control the default item types assigned during import.

Where do I find it?

Customer Default	
Application	Advanced Simulation, Design Simulation
Command Finder	**Customer Defaults**
Location in dialog box	**Teamcenter Integration for NX → Import/Export Assembly → Import**

Command	
Application	Advanced Simulation, Design Simulation
Command Finder	**Import Assembly into Teamcenter**

Adding related CAE parts during export

What is it?

When you export an NX assembly from Teamcenter, and there are related CAE parts, you can now add those CAE models to the export assembly operation. To add the CAE models, select the appropriate option for CAE under **Related Parts** on the **Non Masters/Associated Files** tab in the **Export Assembly** dialog box. The available options are:

- **Simulations, FEMs, and Idealized Parts**
- **FEMs and Idealized Parts**
- **Idealized Parts**

The default setting is **None**.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Command Finder	**Export Assembly in Teamcenter**

Save As enhanced to replace CAE Clone

What is it?

The **Clone CAE Parts** command is removed, as its functionality is now available with the **Save As** command. To clone your CAE parts using the **Save As** command:

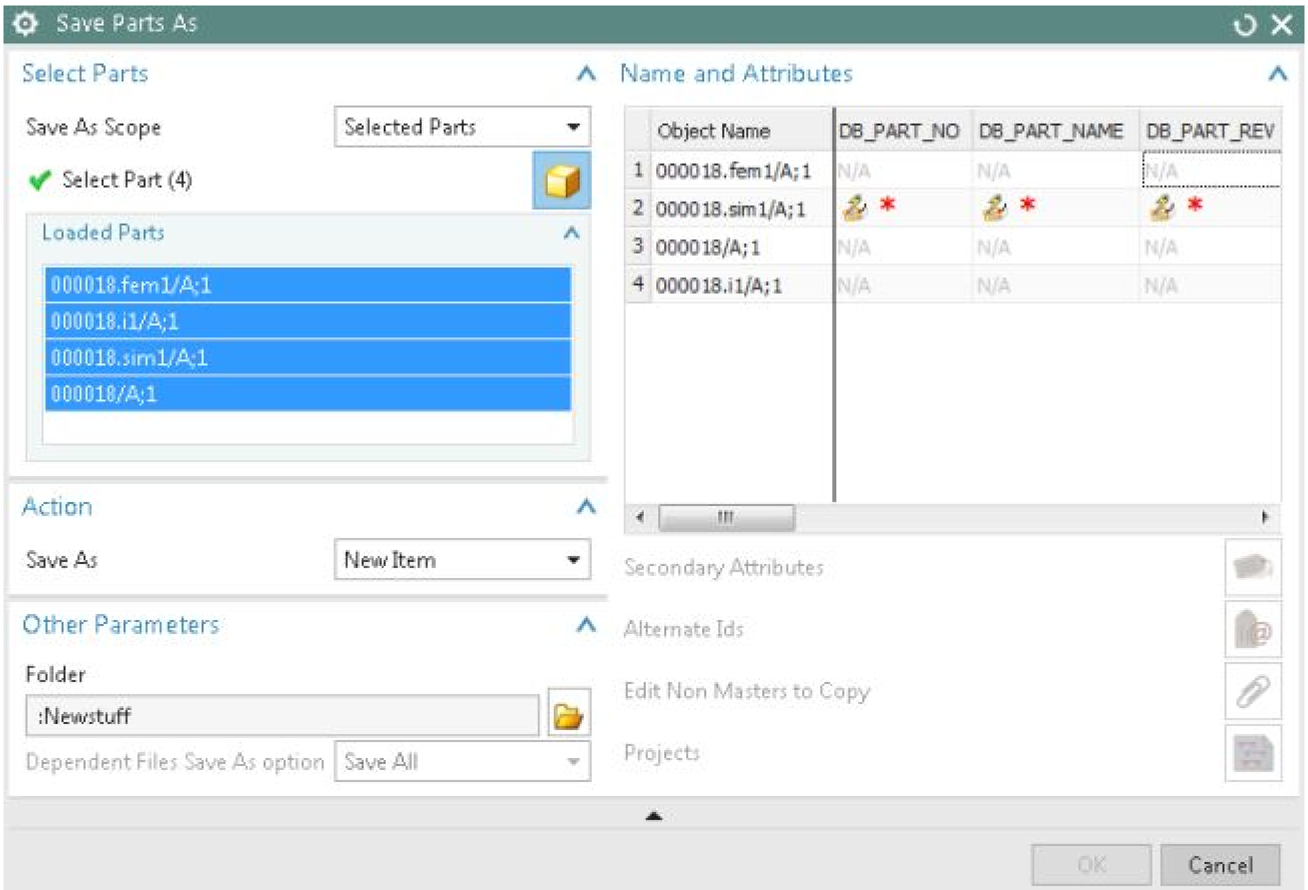
1. Make the part you want to clone the work part.

   - To clone the simulation, make the simulation file the work part.

   - To clone the FEM, make the FEM the work part.

2. In the **Save Parts As** dialog box, set the **Save As Scope** field to **Selected Parts**.

3. Select the parts that you want cloned from the **Loaded Parts** list.



Note

You can generate new items or revisions for the idealized part and master part only if they are fully loaded.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	A FEM or simulation as the work part
Command Finder	**Save As** 

Support for non-master CAE items is removed

What is it?

NX 5 introduced the Simulation Process Management data model. In this data model, NX saves the master part file, idealized part file, FEM file, and simulation file into separate item revisions. Starting with NX 10, you must use the Simulation Process Management data model.

Prior to NX 5, the old Teamcenter data model saved the CAE files as datasets under a master CAD item revision. If you are using the old Teamcenter data model, you must migrate legacy data prior to accessing that data with NX 10. To do this, you can use the `ugnx_caemigrate_tc` command-line utility in NX 9 or an earlier release.

NX Open for CAE

Python

What is it?

The Python programming language is added as a language binding for NX Open. This allows NX users to record and replay NX workflows in the Python language. Advanced users can create or extend NX Open Python journals to develop NX Open applications in Python. These applications can use any of the features of the Python programming language including classes, looping constructs, control statements, and callbacks. Python journals can be recorded and replayed on Windows, Linux, and Mac.

Some of the features provided with NX Open Python include:

- Block Styler support with Python code generation of saved dialog boxes
- Same NX Open API as in the other supported languages
- Wrappers implemented in Python for the NX Open UF API

Why should I use it?

You can use the Python programming language for multi-platform journal recording and replaying in NX.

Where do I find it?

Application	NX Open
Command Finder	Journal

Design Simulation

General capabilities

Identifying objects by color

In the **Simulation Navigator**, the **Color** column lets you view and modify colors of various entities in the graphics window.

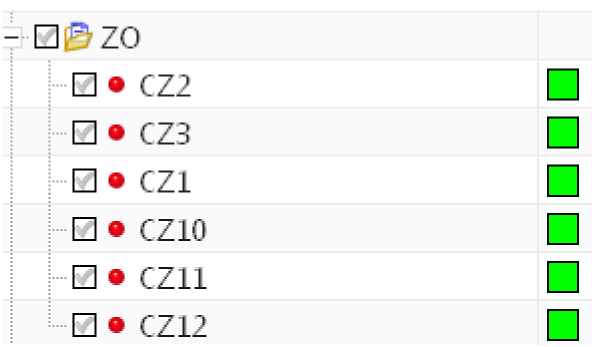
You can view and modify colors for the following entities:

- Polygon bodies
- Meshes
- Boundary conditions

  These include loads, constraints, and Simulation objects.

To modify the color of an object, double-click the color in the **Color** column.

To assign a series of different colors to multiple boundary conditions, select the boundary conditions or a folder containing multiple boundary conditions. Right-click and select **Sequence Display Colors**.



Original colors

Colors after using Sequence Display Colors

To modify the colors in the sequence, in the **Customer Defaults** dialog box, choose **Simulation→Boundary Conditions**, and click the **Sequence Display Colors** tab.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	Boundary conditions applied to the model
Simulation Navigator	Right-click multiple boundary conditions→**Sequence Display Colors**

Solution Monitor enhancements

What is it?

- The **Solution Monitor** is now used by the NX Multiphysics and NX Nastran solvers.

- You can copy and paste the displayed text output. In previous versions, the displayed text output was not selectable.
- The user interface of the **Solution Monitor** is refreshed and the command names are more intuitive. The commands now have tooltips describing their functionality.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	NX Nastran, NX Multiphysics, NX Thermal and Flow, NX Space Systems Thermal, or NX Electronic Systems Cooling as the selected solver environment
Command Finder	**Solve**
Simulation Navigator	Right-click a solution node→**Solve**

Support for non-ASCII characters

What is it?

You can now use multi-byte characters to name files and enter data in the Advanced Simulation, Design Simulation, and Motion applications. For example, you can use multi-byte characters in the names of:

- All Motion objects, such as joints and drivers.
- Solutions, solution steps, or subcases.
- CAE related files, such as FEM and Simulation files.

In NX, your specified locale controls the language used in the user interface as well as the code set for storing character data. For example, if your locale is set to China, you can enter Chinese character data in the NX user interface.

The graphic shows Chinese characters in the **Simulation Navigator**.

NX follows the Unicode international standard to store multi-byte character data, using UTF–8 encoding. Although NX supports the use of multi-byte characters, many third party software packages that interface with NX, such as finite element solvers, support only ASCII character data.

Caution

NX does not provide an error message if the solver receives unsupported characters. You must verify that your solvers will support your desired locale before implementing localization.

Solver support for multi-byte characters

Solver	Unicode support	Solve with non-ASCII entities	Solve with non-ASCII file names	Export with non-ASCII file names and entities	Import with non-ASCII file names and entities
NX Nastran	No	Yes	Yes	Yes	Yes
MSC Nastran	No	No	No	Yes	Yes
Abaqus	No	No	Yes	Yes	Yes
ANSYS	No	No	No	Yes	Yes
NX Nastran Design	No	Yes	Yes	Yes	Yes
NX Multiphysics (**Structural**)	No	Yes	Yes	N/A	Yes
NX Multiphysics (**Thermal**/**Mapping**/**Coupled**)	No	No	No	N/A	Yes
NX Thermal/Flow (**Thermal**/**Advanced Thermal**,**Flow**/**Advanced Flow**, **Coupled Thermal-Flow**, **Axisymmetric Thermal**)	No	No	No	No	Yes
NX Thermal/Flow (**Mapping**/**Axisymmetric Mapping**)	No	Yes	Yes	No	Yes
LS-DYNA	No	Yes	Yes	Yes	Yes
UNV	No	N/A	N/A	Yes	Yes
Shape Optimization	No	No	No	N/A	N/A

Solver	Unicode support	Solve with non-ASCII entities	Solve with non-ASCII file names	Export with non-ASCII file names and entities	Import with non-ASCII file names and entities
Topology Optimization	No	No	No	N/A	N/A
Durability	No	No	Yes	N/A	N/A
Geometry Optimization	No	Yes	Yes	N/A	N/A
Adaptivity	No	Yes	Yes	N/A	N/A
NX Motion Co-Simulation	No	No	No	N/A	N/A
NX Motion Recurdyn	No	Yes	Yes	N/A	N/A
NX Motion ADAMS	No	Yes	Yes	N/A	N/A

Solutions

Making copies of subcases and steps

What is it?

You can now make copies of subcases or steps using the **Clone Multiple** command.

When you use this command:

- The cloned copies include the loads and constraints from the original subcase or step.
- The name of each cloned copy includes the original entity plus a numerical suffix.
- The last copy created becomes the new active subcase or step.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	A Simulation file as the work part and displayed part
Simulation Navigator	Right-click a subcase or step → **Clone Multiple**

Editing boundary condition-step association using a spreadsheet

What is it?

You can now export boundary condition-step associations to a spreadsheet, make changes in the spreadsheet, and then import the associations back into NX. In previous versions, you could export the associations, but you could not import them back into NX.

	A	B	C	D	E
1	Subcase Name	Force(1)	Force(2)	Pressure(1)	Pressure(2)
2	GLOBAL	X	X	X	X
3	Subcase - Static Loads 1	N	N	N	N
4	Subcase - Static Loads 2	20.0000	N	N	N
5	Subcase - Static Loads 3	N	10.0000	50.0000	1.0000
6	Subcase - Static Loads 4	N	N	N	N

With this release, use these steps:

1. To edit your data in a spreadsheet, in the **Step/Subcase Manager** dialog box, right-click in an empty area of the **Step/Subcase List** table and choose **Edit in Spreadsheet**.

2. Make your changes and save the spreadsheet as a comma-separated values (CSV) file.

3. To import your edited file, in the **Step/Subcase Manager** dialog box, right-click in an empty area of the **Step/Subcase List** table and choose **Import from CSV**.

This release also adds the **Export to CSV** command, which lets you export the association data from NX to a file in the comma-separated values format.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	A Simulation file as the work and displayed part At least one active solution that contains multiple steps or subcases
Simulation Navigator	Right-click a selected solution→**Step Manager** or **Subcase Manager** (Nastran)

Boundary conditions

Temperature load enhancements

What is it?

You can now define a temperature load by importing temperature results. You do this using a **Temperature — External Time Unassigned** type of **Temperature** load. You can import constant temperatures from Nastran, Abaqus, ANSYS, and NX Thermal results files.

Corresponding Nastran syntax

The new types of **Temperature** load correspond to different NX Nastran bulk data entries.

- If you select the **Temperature — External Time Assigned** type, NX creates the `DTEMPEX` bulk data entry.

- For all other types, NX creates the `TEMP` bulk data entry.

Where do I find it?

Application	Design Simulation
Prerequisites	Structural as the specified analysis type
Command Finder	**Temperature Load**

Post-processing

General post-processing enhancements

What is it?

This release includes a number of general enhancements to NX Post-processing. These include:

- Support for Kelvin
- Graphing enhancements
- New derived results for worst principal stress

Support for Kelvin

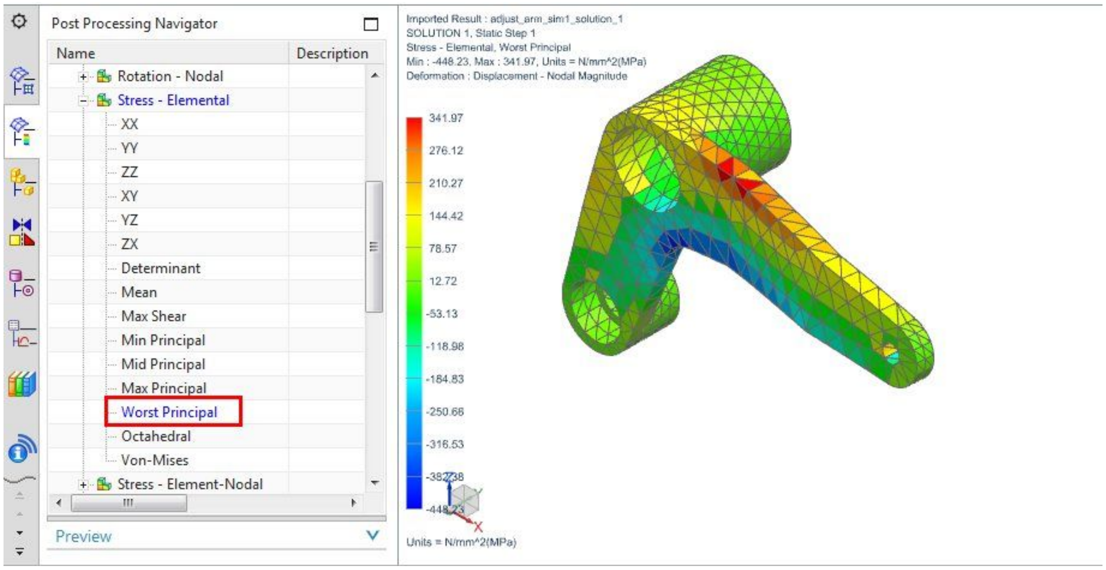
You can now display and manipulate results in Kelvin in NX Post-processing, when you have one or more results files loaded.

Graphing enhancements

You can now select **Include Intersections** in the **Graph** dialog box to create a smooth graph of the changes in data values between points. This new option is available when the graph **Type** is set to **Path** and the **Define by Path** option is selected.

New derived results for worst principal stress

For stress results at nodes on elements, NX Post-processing now derives the worst principal stress tensor component. The worst principal stress is the principal stress component with the largest absolute value.

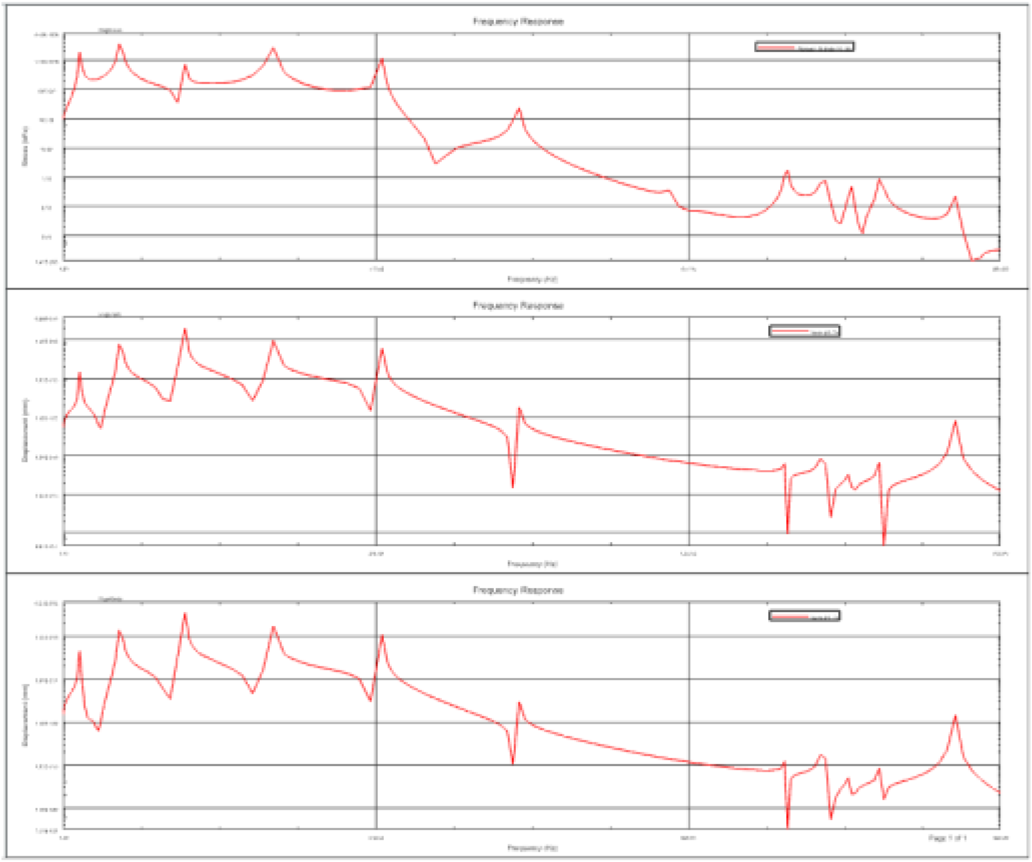


Functions and Graphing

Plotting a stacked graph

What is it?

You can plot multiple functions to multiple subgraphs of a new graph. You do this from the **XY Function Navigator**, using the **Plot(XY) in Stack** command.



Zooming to a specified window in a stacked graph

You can zoom in on the subgraphs to view smaller ranges of data. To do this, use the **Zoom by X Only** and **Zoom by Y Only** commands.

- **Zoom by Y Only** zooms in on only the subgraph that you choose.
- **Zoom by X Only** zooms in on all subgraphs that have the same type of abscissa data.

To expand all subgraphs back to their full ranges, choose **No Window**.

Example:

Suppose you plot a stacked graph where one function has an X range of 0 to 10 seconds, a second function has a range of 0 to 20 seconds, and two additional functions have X values of frequency. If you use the **Zoom by X Only** function to the range of 0 to 3 seconds on either of the time subgraphs, both of them will zoom to that range. The frequency functions continue to show their full range.

Maximum Function Count

You can control how many subgraphs you can create in one graph, using the **Maximum Canvas Count on a Stacked Plot** preference. NX creates the subgraphs starting from the bottom of the screen, and matches the order in which you select the functions. If you select more functions than allowed, a warning appears, and NX plots only the functions that you selected up to the maximum number allowed.

Why should I use it?

You can display multiple functions simultaneously using any one of the following commands: **Plot(XY) in Stack**, **Plot(XY)**, **Plot(XYZ)**, and **Overlay**. The advantage of using the **Plot(XY) in Stack** command is that it is the only one that allows you to include functions that have unique abscissa units, and it allows you to zoom in on a single function.

Where do I find it?

Plot(XY) in Stack

Application	Advanced Simulation
XY Function Navigator	Right-click selected multiple functions → **Plot(XY) in Stack**

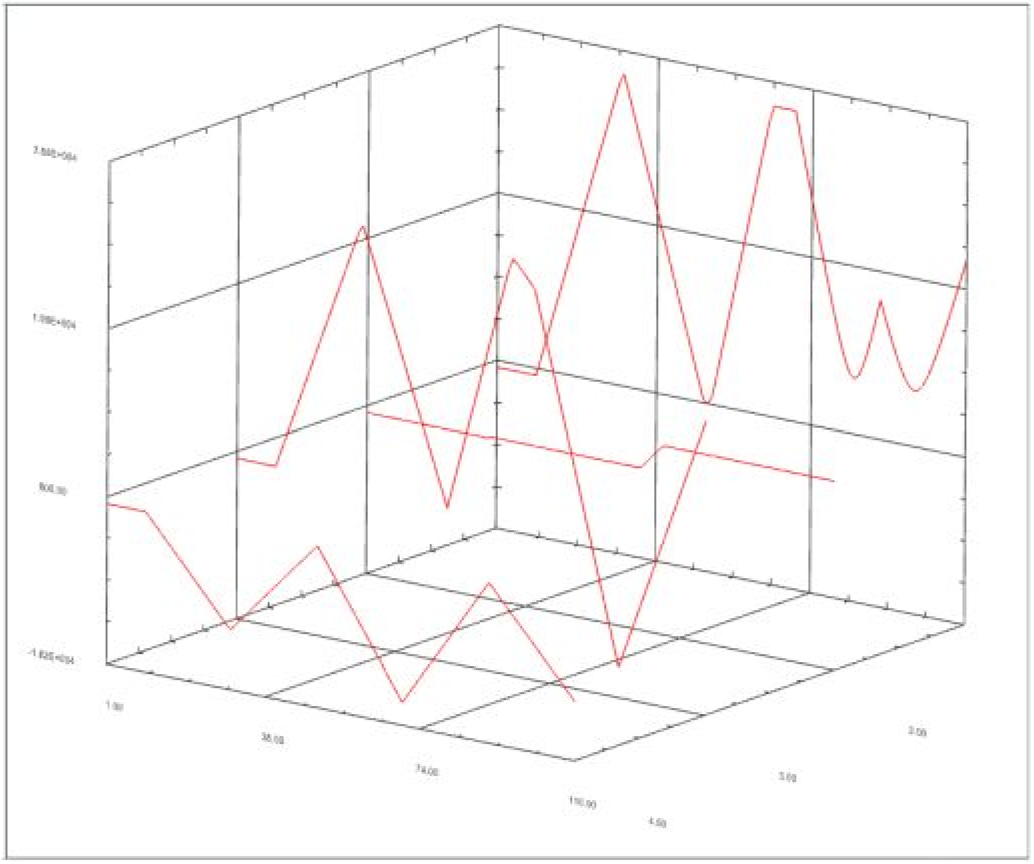
XY Plot Preferences

Menu	**File** → **Preferences** → **XY Plot**
Location in dialog box	**XY Plot Preferences** dialog box → **Maximum Canvas Count on a Stacked Plot**

Plotting 2D functions to a 3D graph

What is it?

You can plot multiple 2D functions to unique XY planes of a 3D graph. You do this using the **Plot(XYZ)** command. Each function you select must have identical data types for every axis.



You select one of the following options from the **Z-Axis Definition** command to define the XY plane on which NX places each function.

Selection Order	Plots the function to a Z value that represents the order in which you selected the functions.
General Value	Uses the **Z General Value** data attribute for each function.
RPM value	Uses the **Z RPM value** data attribute for each function.
Time Value	Uses the **Z Time Value** data attribute for each function.
Order Value	Uses the **Z Order Value** data attribute for each function.

Note

You can only use the **Z-Axis Definition** command to control the placement of AFU records. NX places all other record types in the selection order.

Creating 3D graphs when using the Plot(XY) command

When you plot complex functions using the **Plot(XY)** command, you can create a 3D graph using the **3D Argand** option. NX plots the real and imaginary components of the ordinate to the X-axis and Y-axis respectively, and the abscissa to the Z-axis.

Zooming in on graphs

You can use the new **Zoom by Z Only** command as well as **Zoom by X Only** and **Zoom by Y Only** to zoom into a window of a 3D graph.

Note

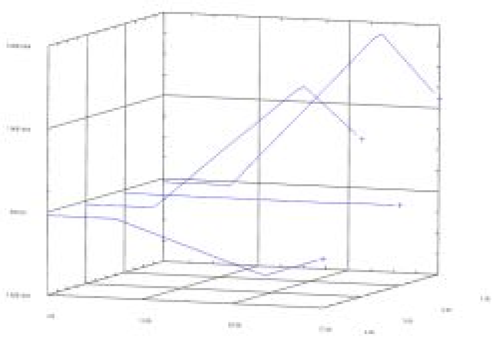
You cannot use the **Zoom by Rectangle** command on a 3D graph. You cannot use the **Zoom by Y Only** command on a 3D graph, if your **Graph Style** is set to **Surface**.

Editing the display options

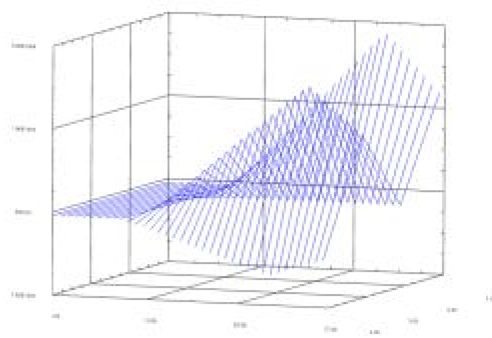
More display options are available for 3D graphs than for 2D graphs. You can choose the **Surface** graph style to connect your functions into a singular surface.

For functions plotted to a **Line** graph style, you can define the direction of the lines.

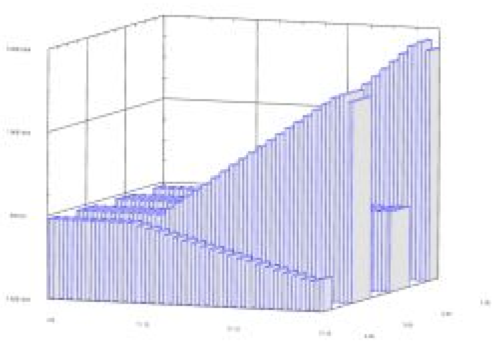
- **X** — Plots the curve as a line that connects the data points on each XY plane.
- **Z** — Plots the curve as a series of lines that connect the data points from each XY plane to the next one.
- **X&Z** — Plots the curve as both a line that connects the data points on each XY plane, and a series of lines that connect the data points from each XY plane to the next one.



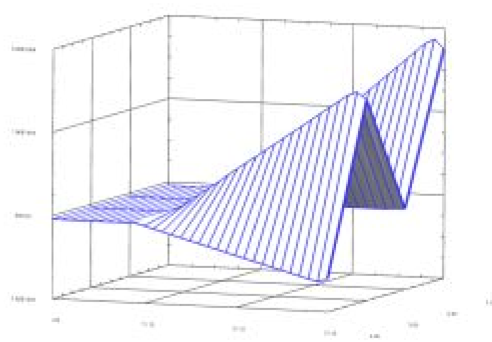
Line graph style, X direction



Line graph style, Z direction



Bar graph style



Surface graph style

Where do I find it?

Application	Advanced Simulation

Prerequisite	Multiple AFU records must be selected
XY Function Navigator	Right-click selected records → **Plot(XYZ)**

Capturing an image from a graph

What is it?

Use the **Capture Image** command to save an image of a plotted graph. You can save your image in the following formats:

- JPG
- TIFF
- PNG

NX saves the image to a directory that you select. To capture your image with a white background, use the **Use White Background** option.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A graph plotted to the main graphics window
Command Finder	**Image Capture**

Save to AFU command renamed

What is it?

The **Save to AFU** command is renamed to **Save Graph**, as you can now save the graph to a CSV file as well as to an AFU file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A graph plotted to the main graphics window
Command Finder	**Save Graph**

Plotting unlimited number of functions to a single 2D graph

What is it?

NX no longer limits the number of records you can plot to a single graph.

To improve performance, there is a **Maximum Legend Count on a Graph** preference. When you create a graph with more functions than the quantity specified by this preference, NX disables synchronized tracking and does not display the graph legend.

Where do I find it?

Application	Advanced Simulation
XY Function Navigator	Right-click a function → **Plot(XY)** → **XY Plot Preferences**
Location in dialog box	**Maximum Count** → **Maximum Legend Count on a Graph**

Editing multiple functions simultaneously

What is it?

You can select multiple functions in the **XY Function Editor** and edit them simultaneously.

The AFU records are listed in the order in which you select them. The first record in the list is initially the active record. A wrench icon indicates the active record. You can edit just the active record or all of the records in the list. Double-click a different record to make it active.

Each field in the **XY Function Editor** is preceded by one of the following icons representing the attributes status:

- indicates that the selected functions have the same value.
- indicates that the selected functions have mixed values.
- indicates that a change will be applied after you enter in a new value.

To modify only the active record, click the **Apply Change — Active Record** button. To modify all of the records that include the modified fields, click the **Apply Change — Applicable Record** button.

Where do I find it?

Application	Advanced Simulation
XY Function Navigator	Right-click multiple selected functions → **Edit**

Additional math operations available for functions

What is it?

The **Single Math** command now includes the following math operations.

Independent Integration	Performs an integration of the function. Ordinate data of type Acceleration becomes Velocity, and of type Velocity becomes Displacement. The number of data points in the function is one less than in the original function.

Independent Differentiation	Performs a differentiation of the function. Ordinate data of type Displacement becomes Velocity, and of type Velocity becomes Acceleration. The number of data points in the function is one less than in the original function.
Frequency Integration	Used only for functions with abscissa data of type Frequency. Performs an integration of the function. Ordinate data of type Acceleration becomes Velocity, and of type Velocity becomes Displacement. The resultant integral is a complex function.
Frequency Differentiation	Used only for functions with abscissa data of type Frequency. Performs a differentiation of the function. Ordinate data of type Displacement becomes Velocity, and of type Velocity becomes Acceleration. The resultant derivative is a complex function.

Where do I find it?

Application	Advanced Simulation
Command Finder	**Single Math** 

Updating axis limits of graphs when using overlays

What is it?

NX can automatically update the limits of a plotted graph when you overlay a new function onto it. To enable this functionality, set the **Update Axis Limits for Overlaying Plot** preference in the **XY Plot Preferences** dialog box. If you do not want an automatic update but would like to apply the update when you need to, use the **Fit Axis Limits** command to update the axis.

If you applied a zoom window to the graph, the axis limits will change but the zoom will continue to be observed. You can use the **No Window** command to update the display to show the new full axis limits.

Where do I find it?

Update Axis Limits for Overlaying Plot

Application	Advanced Simulation
Location in dialog box	**XY Plot Preferences** dialog box → **Overlay** group → **Update Axis Limits for Overlaying Plot** check box

Fit Axis Limits

Application	Advanced Simulation
Prerequisite	A function plotted that exceeds the current axis limits
Command Finder	**Editing**  → Right-click the graph background → **Fit Axis Limits**

Managing graphics windows

What is it?

On Windows machines, there are new options for managing the display of multiple graphics windows. The new **Graphics Window Manager** lists all of your independent graphics windows with options to control their screen position.

Close	Closes the selected window.
Make Floating	Restores a docked graphics window to float independently in the foreground.
Make Current	Docks the selected window in front of the main graphics window.
Make Main Graphics Window Current	Moves the main graphics window in front of any docked graphics windows.

You can also tile or cascade your graphics windows. Use the following commands in the **Window** drop-down list to tile or cascade your separate graphics windows as well as your main graphics window:

- **Cascade**
- **Tile Horizontally**
- **Tile Vertically**

Where do I find it?

Application	Advanced Simulation
Prerequisite	One or more graphs plotted to separate graphics windows
Command Finder	**Graphics Window Manager**

General functions and graphing enhancements

What is it?

- You can maximize the display of independent graph windows.
- You can move the display of a graph from an independent graph window to be superimposed on the NX graphics window. To do this, use the **Send to NX Window** command.
- The **Curve Options** dialog box is now named **Display Options**. The thickness options have increased to include 9 standard thicknesses.

- The spacing of dashed lines is now based on the plot’s dimensions. In previous releases, this was based on the distance between data points, which could make dashed lines appear solid when data points were close together.

Chapter 6: Inspection and validation

Check-Mate

Check-Mate checkers and functions

What is it?

Check-Mate is enhanced to include the new function, mqc_checkWeldPointConnectedParts. You can find it in the *mqc_weld_checker_func.dfa* source file.

The following new Check-Mate checkers are provided with this release.

Checker name	Categories
Number of Hidden Objects	**Get Information→Modeling**
Connected Parts	**Welding→Weld Point**

The following checkers have been enhanced.

Checker name	Categories	Enhancement
Report Solid Body without Density Setting	**Get Information→Modeling**	Adds an option to include sheet bodies.
Report Solid Body with Default Density Setting	**Get Information→Modeling**	Adds an option to include sheet bodies.
Report Solid Body with Specified Density Setting	**Get Information→Modeling**	Adds an option to include sheet bodies.
Report Out-of-Date Features	**Get Information→Modeling**	Adds an option to check internal features.

The following functions have been moved from *mqc_checker_func.dfa* to *mqc_obsolete_func.dfa*.

- mqc_checkViewScale
- mqc_checkViewNames
- mqc_checkCategoryLayer
- mqc_checkLayerCategory
- mqc_checkRequiredCategory
- mqc_checkCategoryObjectMapping
- mqc_checkRefSetsCsysName
- mqc_checkObjectInCategory

- mqc_checkRequiredRefSets
- mqc_checkRefSetsObjectType

The mqc_report_blanked_entity function, which was in the **Get Information→Modeling** category, is now hidden in the **Set Up Tests** dialog box. It is replaced by the mqc_report_hidden_objects function because the obsolete term *blanked entity* has been replaced by the term *hidden object* in NX.

The hidden checker is only invisible in the category list box of the **Tests** tab in the **Set Up Tests** dialog box. If you have already used the hidden checker, you are not affected by this change.

Please see the *ugcheckmate.dat* file in the Check-Mate kit for additional details.

Where do I find it?

Resource bar	**HD3D Tools** →**Check-Mate**

CMM Inspection Programming NX 9.0.1 — NX 10.0

CMM general enhancements

General enhancements — NX 9.0.1

View Vector name change

When you set the tolerance type to **Perpendicularity** or **Parallelism** in the **Tolerance Operation** dialog box, the **View Vector** group is renamed **Tolerance Zone Width Direction**. The new name better reflects the DMIS standard for this vector: "[The] direction vector along which the view dependent parallel [or perpendicular *sic*] plane tolerance zone width is to be applied."

Coordinate tolerance nominal values displayed

To support CMM software that requires DMIS `CORTOL` values to evaluate coordinate tolerances, the **Tolerance Operation** dialog box now shows the nominal X, Y, or Z axis value of the feature center point in the currently aligned Part Coordinate System (PCS).

Improved graphics when defining a PCS from datums

When you define an alignment command in the **Alignment** dialog box using the **Define PCS from Datums** alignment type, NX now indicates which feature is assigned to which letter. In previous releases, NX only highlighted the feature assigned to the datums after you set the alignment direction or origin. The datum feature is now highlighted when you select it from the **Datum Letter** list or type its value in the **Datum Letter** box.

Where do I find it?

Initial release	NX 9.0.1
Application	CMM Inspection Programming

Prerequisite	• To define a tolerance operation, create an appropriate inspection feature. • To create a PCS from datums, create appropriate inspection features and datums from those features.
Command Finder	**Perpendicularity** **Parallelism** **Coordinate Tolerance** **Alignment**

General enhancements — NX 9.0.2

Overriding direct dimension tolerance values defined in PMI

Direct dimension tolerance values defined in a master model may not reflect the design intent. For example, an acceptable angle on the part may be 89.998 degrees instead of the 90 degrees prescribed in a drawing, with acceptable measurement variation causing the angle tolerance to fail. In the **Tolerance Operation** dialog box, in the **Parameters** group, you can override direct dimension **Nominal**, **Upper**, and **Lower** tolerance values linked to PMI.

When you override these values, the **Inherited** icon changes to a **Locally defined** icon to indicate that the PMI value has been overridden. To recall the PMI value, click the unlock icon and choose **Inherited from PMI**.

Collision avoidance progress bar

When you run collision avoidance on inspection paths, a progress bar reports the percentage complete and displays each action.

Performing analysis... 28% **PATH_CYLINDER_TOP - Find Sensor**

Where do I find it?

Initial release	NX 9.0.2
Application	CMM Inspection Programming
Prerequisite	• To define a dimensional tolerance operation, create an appropriate inspection feature. • To apply collision avoidance, in the **Inspection Navigator**, select one or more inspection paths.
Command Finder	**Collision Avoidance**

General enhancements — NX 9.0.3 MP1

New method added to the NX Open API

A **GetRoot** method has been added to the **InspectionSetup** class to return the root of the specified branch root, such as the Machine or Program group name.

Feature selection when adding tolerances

In previous versions, when you added a tolerance to a highlighted feature in the **Inspection Navigator**, the highlight jumped to the newly added tolerance. To add additional tolerances to the feature, you had to highlight the feature again.

The feature now remains highlighted after you add a tolerance. After you add a tolerance to a feature, you no longer have to reselect the feature before you add an additional tolerance. The selection highlight remains on the feature and does not move to the tolerance you create.

Modifying duplicate feature names

In previous versions, if you entered a duplicate inspection feature name in a feature dialog box, NX:

1. Displayed a message stating that the name was already in use.
2. Closed the feature dialog box when you closed the message.
3. Applied a default name to the feature.

To change the default name, you had to double-click it in the **Inspection Navigator** and enter your preferred name.

After you close the message, the feature dialog box now remains open with the duplicate name intact so that you can modify the text to create a unique name.

Pattern features

You can now pre-select features in the **Inspection Operation Navigator** to create a pattern feature. For example, you can select all hole features in your model. NX populates the feature list with the selected features.

You can modify the feature list before you complete the pattern. If you select other operation types by mistake, NX does not add them to the feature list.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	CMM Inspection Programming

Path statuses in the Inspection Navigator

What is it?

The **Path** column in the **Inspection Navigator** now provides additional status information.

Icon	Meaning

- Next to an inspection feature, this icon indicates that there is no path or construction for the feature.

- Next to an inspection path, this icon indicates that the path contains no measurements.

Next to an inspection path, this icon indicates that:

- The total number of points in the path is greater than zero but less than the minimum number of points specified for the feature type in the DMIS 5.2 standard.

- The path consists of a single arc scan on a cylinder, cone, or torus.

- The path consists of a single line scan on a plane, cylinder, or cone.

Why should I use it?

Review the **Path** column to quickly identify potential program problems.

Where do I find it?

Initial release	NX 9.0.1
Application	CMM Inspection Programming
Prerequisite	You must create inspection features and paths.
Inspection Navigator	**Path** column

DMIS Equator and mirroring postprocessors

What is it?

Special postprocessors are available, including one for use with the Equator 300 included in the machine library.

There are also three DMIS 3.0 postprocessors that work with CMM Inspection Execution to mirror coordinates and vectors from symmetrically opposite parts such as two car doors.

- **DMIS 3.0 Mirror XZ Plane**

- **DMIS 3.0 Mirror YZ Plane**

- **DMIS 3.0 Mirror XY Plane**

Versions of these postprocessors for NX 8.5.3 and NX 8.5.2 are also available from GTAC.

Why should I use it?

Mirroring DMIS content reduces the amount of work required to apply tolerances to symmetrical sides of a part or assembly.

Where do I find it?

Initial release	NX 9.0.1
Application	CMM Inspection Programming

Prerequisite	You must define mirrored sensors in your inspection program.
Command Finder	**Post Process** 

Tip type and tool selection

What is it?

A **Sensor Tip Type** list has been added to the **Sensor** group in many dialog boxes. From the **Inspection Method** dialog box, this list allows you to specify a preferred sensor tip for each type of inspection feature. When you create or generate inspection path sub-operations, CMM Inspection Programming selects the preferred tip for each feature type by default.

Sensor tip types include the following.

- **Any** in the **Inspection Method** dialog box allows CMM Inspection Programming to determine the appropriate tip for the feature type, in accordance with the default or selected feature type's sensor strategy. When selected in the **Link PMI** and **Create Paths** dialog boxes, the application uses the preferred tip from the feature methods.

- **Sphere** uses a spherical tip.
- **Cylinder** uses a cylindrical tip for sheet metal edge and surface measurements.

Why should I use it?

Setting an appropriate default sensor type for each type of inspection feature in the **Inspection Method** dialog box is faster than setting or resetting the sensors manually in each inspection path sub-operation.

Where do I find it?

Initial release	NX 9.0.1
Application	CMM Inspection Programming
Prerequisite	You must load the probes specified in your inspection methods.
Location in dialog box	**Sensor** group in the following dialog boxes: • **Inspection Method** • **Create Sub-Operation** • **Link PMI** • **Create Paths**

Sub-operation inheritance from feature methods

What is it?

Inspection paths created using the **Link PMI** and **Create Paths** dialog boxes now inherit sub-operation types from inspection feature methods.

Feature Type	Supported Sub-operation types
Cylinder	**Point Set**, **Scan Helix**
Circle	**Point Set**, **Scan Arc**, **Scan Curve**, **5–axis Scan Curve**
Arc, line	**Point Set**, **Scan Arc**, **Scan Curve**, **5–axis Scan Curve**
Curve	**Point Set**, **Scan Curve**, **5-axis Scan Curve**

Note

- If the default sub-operation type in the method is set to **Scan Arc** or **Scan Helix**, the methods's **Scan Mode** value determines whether the path uses 3-axis or 5-axis motion.
- If the default sub-operation type is **Scan Line**, **Scan Curve**, or **5-axis Scan Curve**, the **Scan Mode** value in the method is ignored.

Why should I use it?

When you specify a default sub-operation and scan mode value for each inspection feature's method, you can quickly generate inspection paths with the appropriate sub-operations from the **Link PMI** and **Create Paths** dialog boxes.

Where do I find it?

Initial release	NX 9.0.1
Application	CMM Inspection Programming
Inspection Navigator	**Home** tab→Top Border bar→**Method View** →double-click the feature method

Collision avoidance enhancements

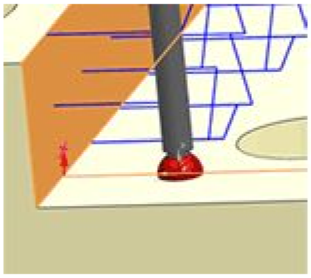
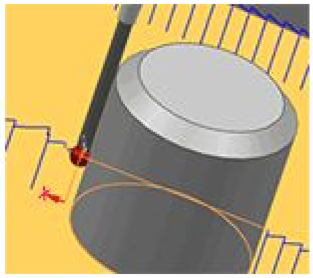
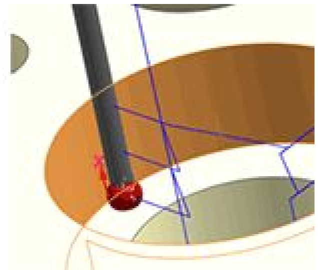
What is it?

Two groups in the Collision Avoidance dialog box are enhanced.

Avoidance Methods group

Two options are added to the **Avoidance Methods** group.

- **Move Measurement Point** moves measurement points to safe locations on plane, cylinder, and cone inspection features if they are causing collisions because they are too close to other geometry.

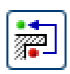
- **Delete Measurement Point** deletes measurement points from any inspection feature if no avoidance solution can be computed.

Avoidance methods have also been reordered in the **Collision Avoidance** dialog box to better reflect the order in which they are logically executed.

Collision Detection Settings group

You can now directly modify **Maximum Length Increment**, **Maximum Angular Increment**, and **Faceting Tolerance** parameters, and they will correctly represent the values used in collision avoidance. In previous releases, you had to set these values in other dialog boxes, and the values did not always reflect the values used in collision avoidance.

Where do I find it?

Initial release	NX 9.0.1
Application	CMM Inspection Programming
Prerequisite	You must select at least one inspection path in the **Program Order** view of the **Inspection Navigator**.
Command Finder	**Collision Avoidance** 
Inspection Navigator	Right-click the path→**Collision Avoidance**

Data analysis options

Ability to save loaded analysis data

What is it?

You can now access your analysis data from **Group** folders in the **Part Navigator**. You must first load the data from one or more measurement (.mea or .dml) files to a layer and give your data a group name that helps you identify your data in the **Part Navigator**.

Why should I use it?

You can now save and organize data from multiple data files to:

- Compare differences in measurement results.
- Use NX measurement tools to display deviations and create annotations for reports.
- Delete problem measurement points that cause tolerances to fail.

Where do I find it?

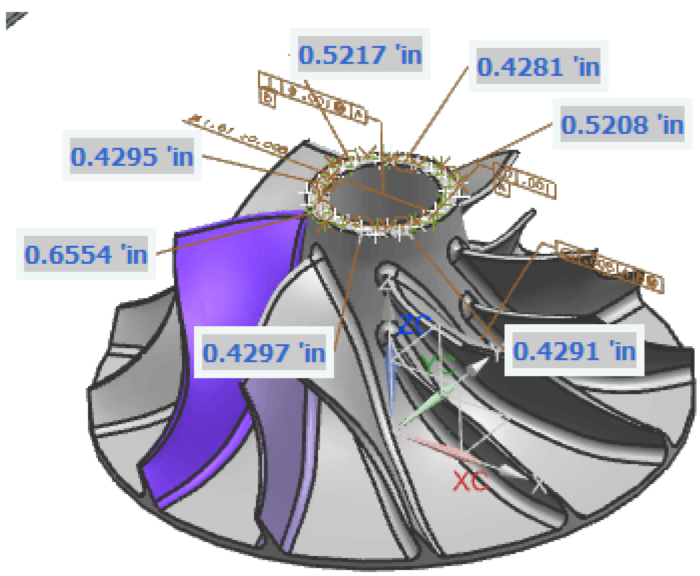
Initial release	NX 9.0.1
Application	CMM Inspection Programming
Prerequisite	You must load an .mea or .dml file and in the **Inspection Navigator**, select the inspection features and tolerances for which you want to save data.
Command Finder	**Save Analyze Data to Layer**
Inspection Navigator	Right-click a selected area and choose **Save Analyze Data to Layer**

Creating annotations for data analysis

What is it?

When you save analysis data to an inspection program, measurement points and lines become accessible from the **Part Navigator**. You can:

- Select features or tolerance groups in the **Part Navigator** to identify deviations between nominal points and their actual measured locations.
- Use NX measurement commands to measure the deviations and create annotations for reports.



Why should I use it?

Annotations support the visualization of problems and large deviations from nominal values in your programs.

Where do I find it?

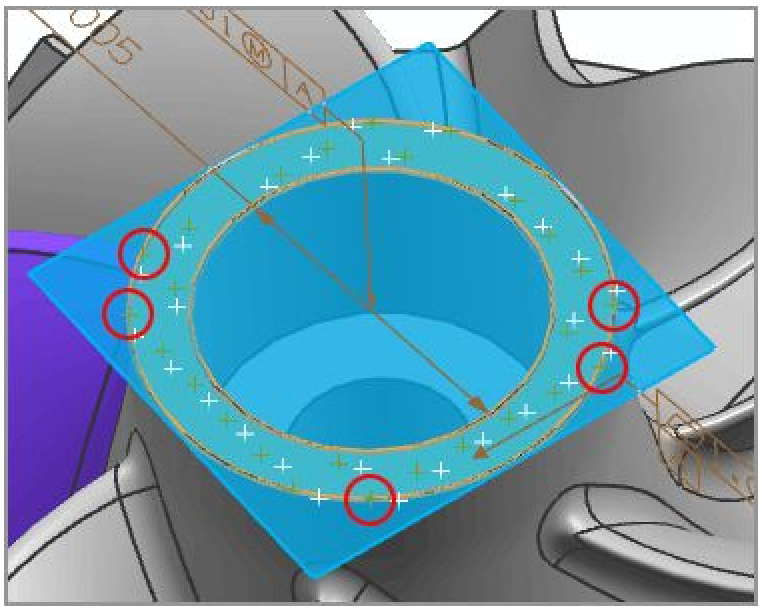
Initial release	NX 9.0.1
Application	CMM Inspection Programming

Prerequisite	You must save analysis data from an .mea or .dml file to your inspection setup program.
Ribbon bar	**Part Navigator** tab →**Groups** folder

Deleting problematic analysis measurement points

What is it?

You can now delete problematic analysis measurement points that are incorrectly causing tolerances to fail. To do this, you can select the points either in the **Inspection Navigator** or the graphics window, and then use the **Delete Analyze Points** command.

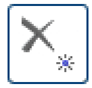


Why should I use it?

Some analysis points may be outliers or lie too close to an edge and cause false readings because they are causing probe collisions with nearby geometry. When you delete such points, you improve statistical accuracy.

Where do I find it?

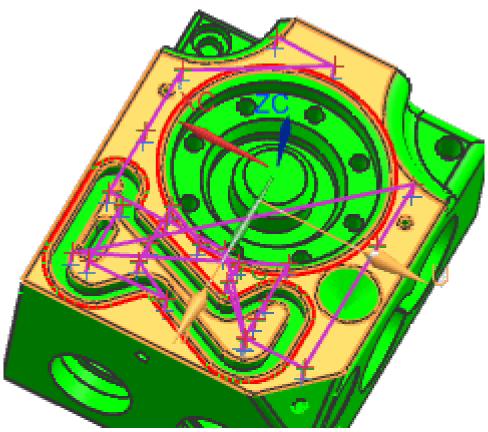
Initial release	NX 9.0.1
Application	CMM Inspection Programming
Prerequisite	You must load measurement data and save that data to a **Part Navigator** layer. In the **Inspection Navigator**, you must select the inspection feature that contains the points that you want to delete.

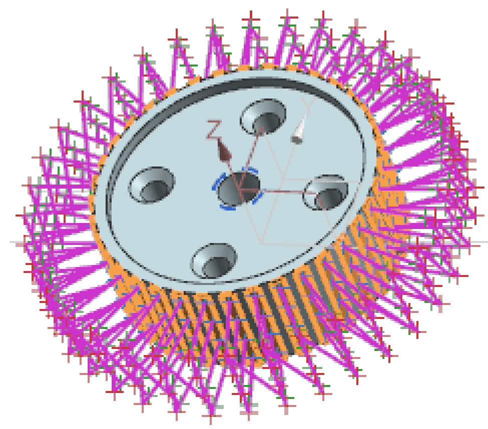
Command Finder	**Delete Analyze Points** 
Inspection Navigator	Right-click the inspection feature that contains the points that you want to delete and choose **Delete Analyze Points**.

Creating plane and cylinder features from multiple faces

What is it?

You can create a single plane feature from multiple coplanar faces, and a single cylinder feature from multiple coaxial surfaces that together form a partial cylinder. When you define a point set sub-operation for either feature in the **Inspection Path** dialog box, each face gets a separate UV grid.





The same logic applies if a tolerance is linked to multiple coplanar faces or coaxial surfaces in PMI. When you link to PMI, the faces or surfaces are generated as single features, and a single inspection path is applied.

Why should I use it?

When you create single plane and cylinder inspection features from multiple faces, you avoid the need to create multiple general surface features and individual inspection paths for those features.

Where do I find it?

Application	CMM Inspection Programming
Command Finder	Plane, Cylinder
Location in dialog box	Creating a single plane feature from multiple coplanar faces: **Plane Inspection Feature** dialog box→**Geometry** group→**Select Plane or Base Point** Creating a single cylinder feature from multiple coaxial faces: **Cylinder Inspection Feature** dialog box→**Geometry** group→**Select Cylinder or Base Point**

Specifying user-defined events (UDEs)

What is it?

You can specify user-defined events (UDEs) to create dialog boxes that let you quickly insert custom code into your post-processed DMIS inspection programs. If you use UDEs, you do not have to write the same code in each program.

When you post process your program, the manufacturing output manager (MOM):

1. Looks for event parameters generated from UDE dialog boxes. Dialog box parameters are defined in a *ude.cdl* file.

2. Handles these event parameters to output custom DMIS code. Event handlers are defined in a *ude.tcl* file.

ude.cdl→*ude.tcl*→Custom DMIS code

Why should I use it?

A custom code event might include a command that CMM Inspection Programming does not support, or a complex function that otherwise takes too much work to define for every inspection program. UDEs make custom code reusable.

Where do I find it?

Initial release	NX 9.0.2
Application	CMM Inspection Programming
Prerequisite	Define your dialog boxes in the *ude.cdl* file and event handlers in the *ude.tcl* file, using a text editor. Restart NX to insert them into your inspection programs.
Inspection Navigator	Right-click the object before which you want the event to occur→**Object**→**Start Events** or **End Events**

Inspection Navigator default object name improvements

The default names and structure of objects in the **Inspection Navigator**, including inspection features, paths, constructed and extract features, and tolerances, are improved.

Inspection feature names

In previous versions, all default inspection feature names used the following string:

INSP_FEAT_<feature type>_<sequence number>

<sequence number> was blank for the first feature of the same type, and began with **_1** for the second.

The string is now changed to:

<feature type><sequence number>.

The sequence number begins at **1** for each feature type.

Example

Inspection path names

In previous versions, inspection paths manually defined in the **Inspection Path** dialog box followed this default format:

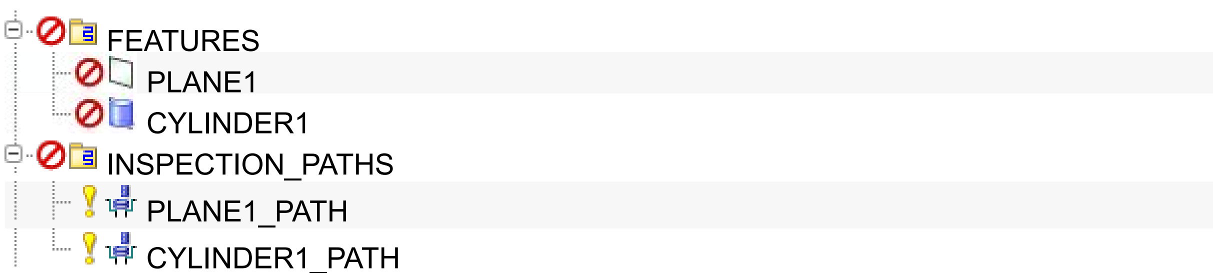
PATH_<feature name>

Path operations created using multi-feature path creation followed this format:

<feature name>_PATH

The path names now follow the latter format regardless of which command you use to create the paths.

Example



NX sets the operation name in the **Inspection Path** dialog box to this format every time you select the feature, and overrides any previous name you entered. You can modify it after it is set and NX retains these modifications unless you select a different feature in the dialog box.

Constructed feature and extracted feature names

In previous versions, the default **Operation Name**:

- In the **Constructed Feature** dialog box was `INSP_CONST`.

- In the **Extract Feature** dialog box was `INSP_EXTRACT`.

Constructed feature and extract feature operation names now derive from the name in the dialog box's **Design Feature Name** text box, with the text strings **_CONST** or **_EXTRACT** appended. The name of the constructed or extract feature updates if you modify the **Design Feature Name**.

Example

Name of an existing feature selected from the **Select Design Feature** list in the **Extract Feature** dialog box if the **Design Feature Mode** = **Select Existing**, or a manually defined name if the **Design Feature Mode** = **Create New**: **MY_PT1**	Constructed feature name: **MY_PT1_CONST**
Name of an existing feature selected from the **Select Design Feature** list in the **Extract Feature** dialog box: **TOP_PLANE1**	Extract feature name: **TOP_PLANE1_EXTRACT**

Tolerance operation names

Default tolerance operation names now use the following format:

<feature name>_<dmis tolerance token string>

Example

If a feature is named **CYLINDER1** and a diameter tolerance is created, the tolerance is by default named **CYLINDER1_DIAM**.

DMIS tokens include the following.

ANGL	CPROFL	FLAT	RAD
ANGLB	CPOFS	PARLEL	STRGHT
ANGLR	CRNOUT	PERP	SYM
CIRLTY	CYLCTY	POS	TRNOUT
COMPOS	DATDEF	PROFL	WIDTH
CONCEN	DIAM	PROFP	
CORTOL	DISTB	PROFS	

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	CMM Inspection Programming

MCOSMOS postprocessor (10.0.1)

What is it?

MCOSMOS, the Mitutoyo Controlled Open Systems for Modular Operation Support postprocessor, is now available as part of the standard installation for NX 10.0.1. It is not necessary to download the files from GTAC.

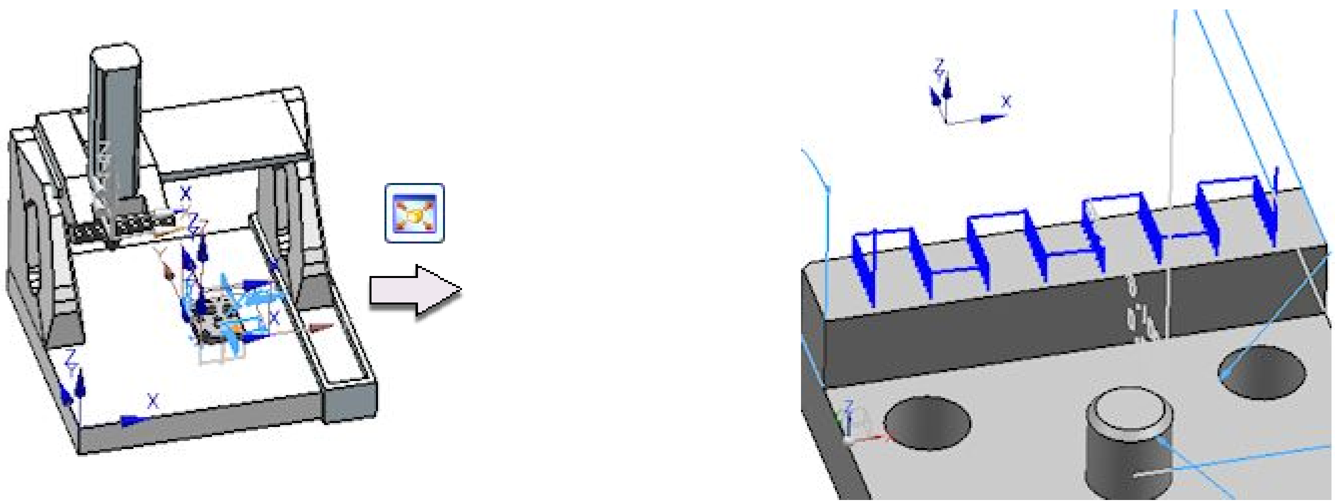
Where do I find it?

Application	CMM Inspection Programming
Command Finder	Postprocess
Location in dialog box	**Postprocess** dialog box→**Postprocessor** group→**MITUTOYO MCOSMOS**

CMM general enhancements (10.0.1)

What is it?

You can fit one or more selected objects to the graphics window view.



Where do I find it?

Application	CMM Inspection Programming
Prerequisite	Select the objects in the **Inspection Navigator**.
Top Border bar	**View** tab→**Orientation** group→**Fit View to Selection**

Chapter 7: Tooling Design

General enhancements

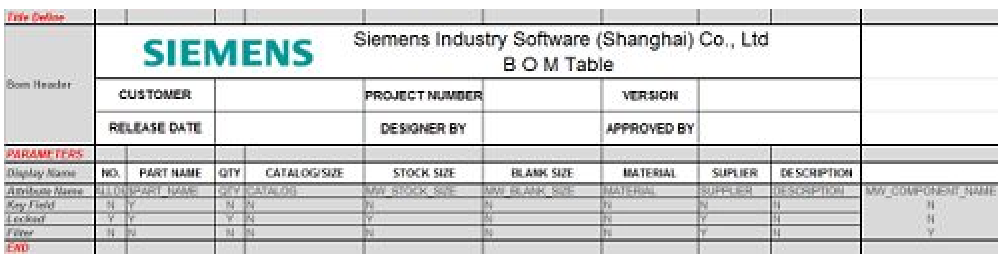
Bill of Material enhancements

What is it?

The existing **Bill of Material** command is enhanced.

Using templates

You can use templates to define the related part information.



You can define the following parameters in the template.

Display Name Specifies the display names of the BOM list items which are displayed as column titles in the **Bill of Material** dialog box.

Attribute Name Specifies the attribute name of the BOM list items. Only the attribute values that you specify in the template are displayed in the BOM list. If you do not specify a value for an attribute, the attribute will not be shown in the BOM list.

Key Field Lets you specify if the corresponding attribute is to be used as a key field attribute or not. The components which have the same key field are grouped together and shown in one row in the BOM list.

Locked Specifies if you can modify the value of an attribute in the BOM list.

Filter Specifies if you want to use the value of an attribute as a filter string.

For example, if the Display Name attribute has a string value set to SUPPLIER, then the attribute will be shown in the BOM list and you can also use the attribute value to filter the BOM data.

If you do not define the Display Name for the filter attributes, for example the MW_COMPONENT_NAME in the template, the attribute will not be shown in the BOM UI.

Note

The cells shown in gray cannot be used as they will be deleted when you export the BOM. You can enter user information at the top of the template.

Changing values

You can change the value from the predefined alteration list in the specified column by defining a common data template. There are four types of lists in common data template, and you can define them by setting LIST_TYPE to values as shown in the following table.

List Type	Output
0	The BOM list with the parameters is displayed in the dialog box. No comparison will be executed.
1	The BOM list with the parameters is displayed in the dialog box, and you can enter information in a string box as well. No comparison will be executed.
2	The BOM list with the parameters is displayed in the dialog box, and if the value of a cell is not in the BOM list, that particular cell will be highlighted.
3	A string of the BOM is exported instead of displaying the BOM list in the dialog box. You can enter information in a cell. If the values that you enter are found in the list then that particular cell is highlighted.

Other enhancements

You can now also do the following:

- Generate different BOM lists by switching templates in the **Bill of Material** dialog box.
- Export the data in a particular row in the BOM list. To do this, select the check box at the beginning of the row.

- Quickly recognize modified information in the BOM list.

  If you modify a row in the BOM list, the particular row is highlighted in blue.

- Filter BOM data based on specific attributes which are defined in the BOM template.

Why should I use it?

These enhancements help you to generate BOM with a formatted bill of material template in Excel. You can specify different part attributes in different columns of the template, specify one or more part attributes to be a key field, a lock status, or to filter the column.

Where do I find it?

Application	Mold Wizard, Progressive Die Wizard, and Electrode Design
Command Finder	**Bill of Material**

Added functionality in Face Color Management

What is it?

- Color bitmaps identify the colors and attributes for the current part. The information is drawn from your specified color spreadsheet. The bitmaps are available in a separate column in the **Face Color Management** dialog box.

- Colors and attributes can be applied to non-circular holes.

- A new **Select Sub-Hole Face Automatically** option is available to help you select complex holes more efficiently.

- The **Show Face Without Any Manufacturing Information** and **Show Face With Same Manufacturing Information** options are available as buttons instead of check boxes.

Why should I use it?

Because colors are associated with manufacturing methods, the ability to assign colors to objects is a valuable way to provide instructions for downstream applications, including drawings.

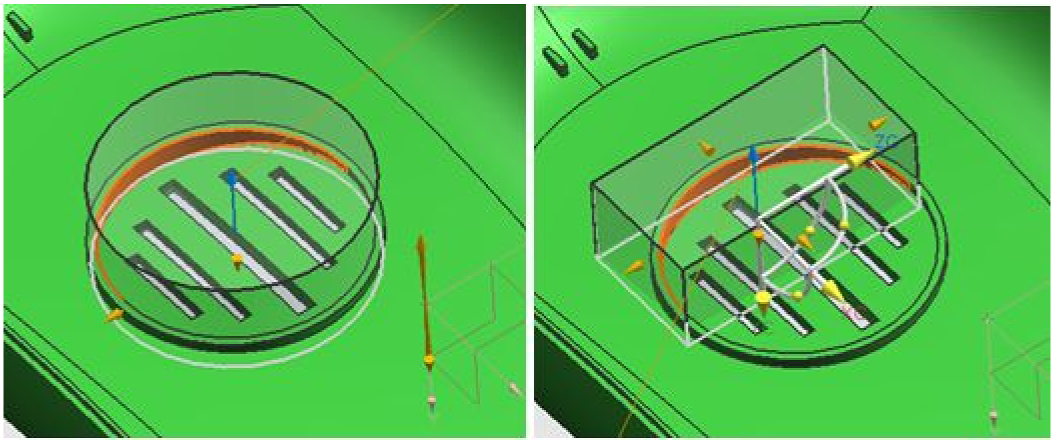
Where do I find it?

Application	Mold Wizard, Progressive Die Wizard, Electrode Design
Command Finder	**Face Color Management**

Electrode Design

Create Box

Use this command to create a block or cylinder which envelopes the selected objects. This command replaces the previous box feature based on Knowledge Fusion.



You should use this command when you want to create a block or a cylinder which envelopes the contour with selected faces, edges, curves, points, solid bodies or facet bodies.

Where do I find it?

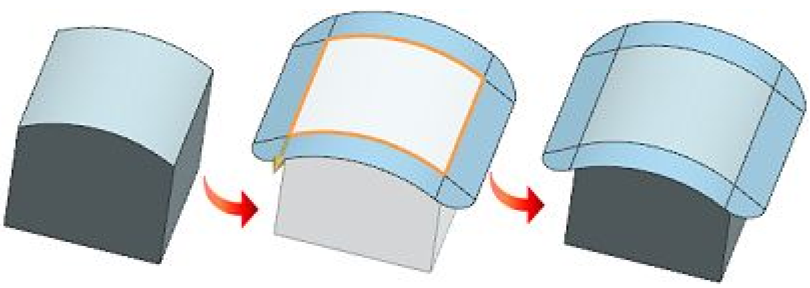
Application	Mold Wizard, Progressive Die Wizard, and Electrode Design
Command Finder	**Create Box** 

CAM Data Preparation

Guided Extension enhancements

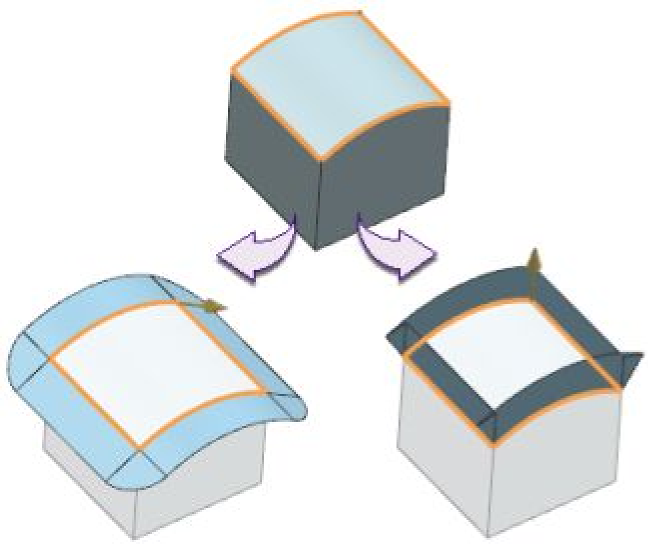
What is it?

You can now also select the edges of a solid body to create surface extensions. In previous releases, you could only select a sheet body.

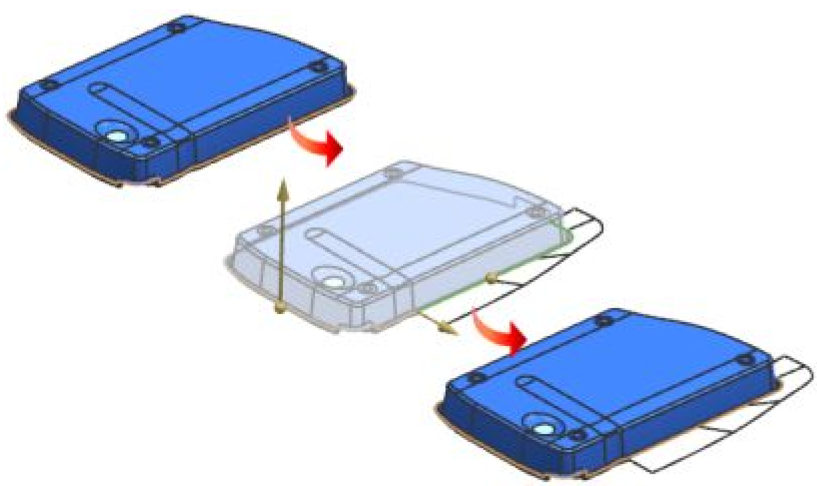


You can:

- Extend the edges of a solid body tangentially to the selected face and select the direction in which you want to create the surface extension. To do this, in the **Type** group, select **Tangent to Face** and click **Change Faces Side** .



- Extend the curves or edges of a solid body or sheet body perpendicular to a specified vector. To do this, in the **Type** group, select **Vector**.



You can also edit the surface extension you create. To do this, double-click the Guided Extension feature in the **Part Navigator**.

- Model History
  - Linked Datum Coordinate System (0)
  - Linked Body (1) “UM_PROD_BODY”
  - Work Piece(2)
  - Linked Body (3) “UM_INSERT_BOX”
  - Feature Group (8) “patch_set1”
  - Extracted Region (9) “CAVITY_EXTRACT_REGION”
  - Extracted Region (10) “CORE_EXTRACT_REGION”
  - Guided Extension (11)

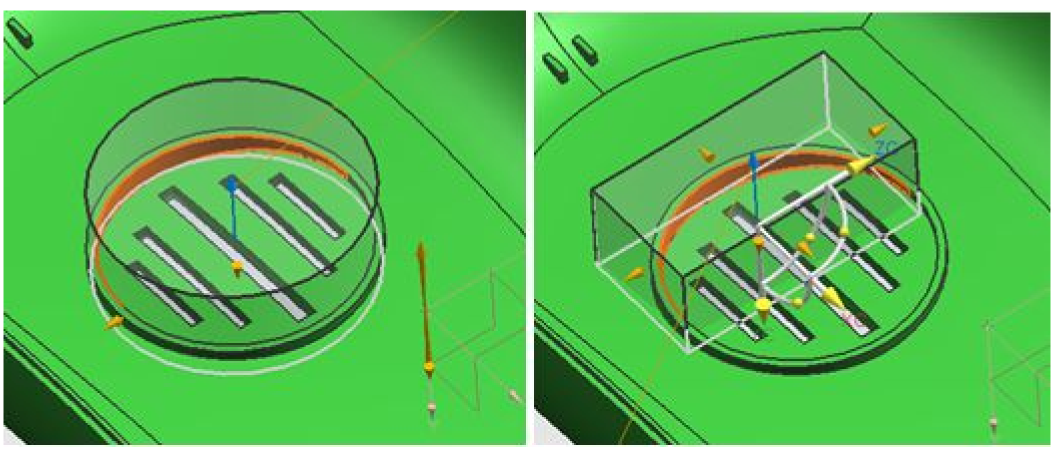
Where do I find it?

Application	Modeling
Prerequisite	The **Change Faces Side** option is available only when you set **Type** to **Tangent from Face**.
Command Finder	**Guided Extension**
Location in dialog box	• **Type** group→Type list→**Tangent from Face** or **Vector** • **Base Profile** group→**Change Faces Side**

Mold Wizard

Create Box

Use this command to create a block or cylinder which envelopes the selected objects. This command replaces the previous box feature based on Knowledge Fusion.



You should use this command when you want to create a block or a cylinder which envelopes the contour with selected faces, edges, curves, points, solid bodies or facet bodies.

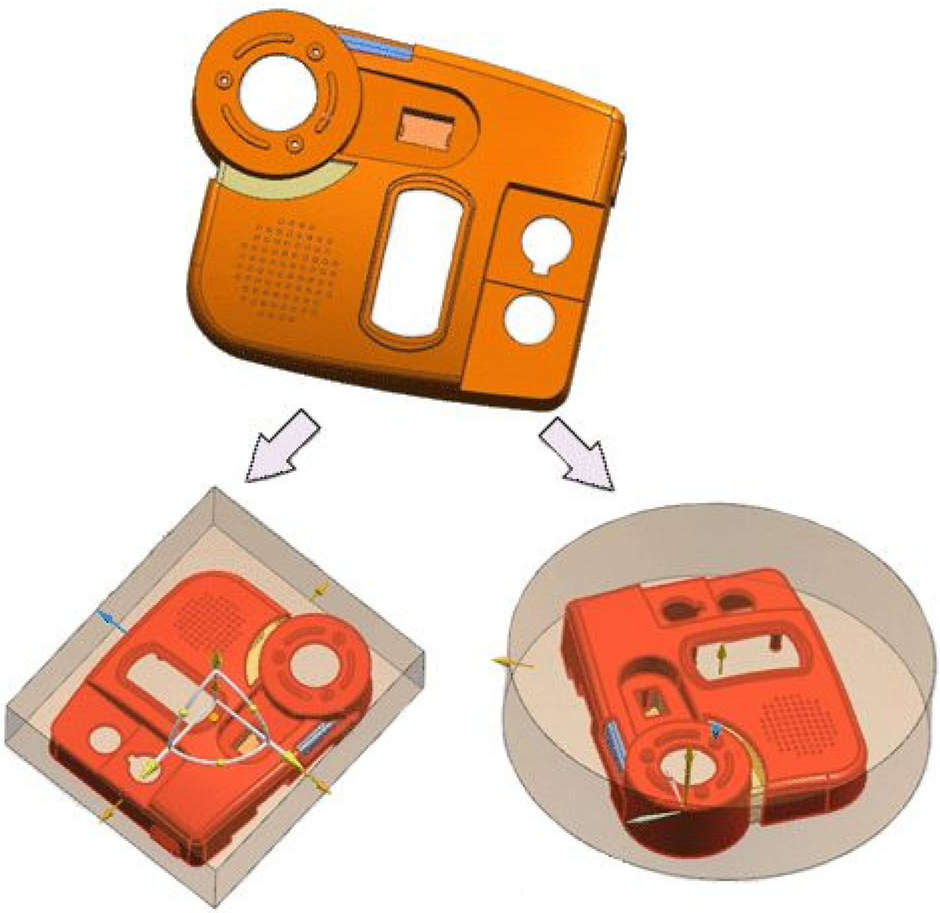
Where do I find it?

Application	Mold Wizard, Progressive Die Wizard, and Electrode Design
Command Finder	**Create Box**

Stock Size enhancements

What is it?

Stock Size is enhanced with a smart object in NX. It now allows being modified with six clearances in six directions. It also provides an option to set zero a clearance value automatically if there are extreme faces in selected body in the specified direction.



Why should I use it?

You should use it when you want to calculate the stock (block or cylinder) size which envelopes the contour with selected solid bodies.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**Stock Size**

Standard parts reuse enhancement

What is it?

You can now browse and select standard parts in the **Reuse Library** when you use the following commands:

- **Standard Parts**
- **Mold Base Library**
- **Slide and Lifter Library**
- **Sub-insert Library**
- **Cooling Component Part Library**

In previous releases, the standard parts were displayed within these dialog boxes which made the dialog boxes unwieldy. Now these dialog boxes are smaller and take up less design space as the standard parts are displayed in the **Reuse Library**.

To open the standard parts in the **Reuse Library**, you can click **Select Item** in the dialog box of the respective command. After you select a part from the **Reuse Library**, its parameters are populated in the respective dialog box. You can change these parameters before you place the standard part in your design.

Where do I find it?

Standard Parts command

Application	Mold Wizard
Command Finder	**Standard Parts**
Location in dialog box	**Standard Part Management** dialog box→**Select Item**

Mold Base Library command

Application	Mold Wizard
Command Finder	**Mold Base Library**
Location in dialog box	**Mold Base Library** dialog box→**Select Item**

Slide and Lifter Library command

Application	Mold Wizard
Command Finder	**Slide and Lifter Library**

Location in dialog box	**Slide and Lifter Design** dialog box→**Select Item**

Sub-insert Library command

Application	Mold Wizard
Command Finder	**Sub-insert Library**
Location in dialog box	**Sub-insert Design** dialog box→**Select Item**

Cooling Component Part Library command

Application	Mold Wizard
Command Finder	**Cooling Standard Part Library**
Location in dialog box	**Cooling Component Design** dialog box→**Select Item**

Motion Simulation commands

What is it?

The **Tooling Motion Simulation** command is removed.

Use the following commands to set up the kinematic model, define the cam, define the lifter, define user defined motion, and run the simulation.

Motion Preprocess

Use this command to set and load the kinematic model, and mount the sheet metal or plastic parts of the die or the mold on the kinematic model.

Why should I use it?

You can use this command to prepare your motion simulation data.

You can:

- Clone a predefined kinematic model for tooling into your current assembly or a directory you specify.
- Change the kinematic model. NX displays a message window to confirm whether you want to remove the current kinematic model and add another one. If you click **Yes**, you must mount the components again and redefine the cams.
- Generate control data and import it to a kinematic model according to the die settings. You can also change to different control data and re-read the control data.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard

Command Finder	**Motion Preprocess**

Define Cam

Use this command to create, edit, or delete a linear cam, a rotary cam, a rocker cam, a user-defined cam, or cushion programming. You can select solid bodies, sheet bodies, or faceted bodies that have solid or sheet bodies associated with it, and use them to define the cam.

Why should I use it?

A predefined kinematic model contains kinematic data about basic cam actions. You can use this command to define the kinematic data for various types of cams. NX automatically calculates and stores the cam data in the kinematic model.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**Define Cam**

Define Lifter

Use this command to create, edit, or delete a generic lifter or a lifter on a lifter. You can select solid bodies, sheet bodies, or faceted bodies that have solid or sheet bodies associated with it, and use them to define the lifter.

Why should I use it?

A predefined kinematic model contains kinematic data about basic lifter actions. You can use this command to define the kinematic data for various types of lifter actions. NX automatically calculates and stores the lifter data in the kinematic model.

Where do I find it?

Application	Progressive Die Wizard and Mold Wizard
Command Finder	**Define Lifter**

User Defined Motion

Use this command to add user defined motion data to your kinematic model.

Why should I use it?

You can define:

- Linear movement along a specific vector.
- Angular movement about a specific axis.

The motion curve is a set of 360 values, one for each timing angle 0-359, that determines how far to move along or about the axis at each point of the simulation. You can use the simple motion curves created by NX, or supply your own values using a comma separated values (CSV) file.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**User Defined Motion**

Run Simulation

In Progressive Die Wizard, use this command to view an animation that shows the kinematics of all the automatic and user-identified components in the model.

Why should I use it?

You can:

- View the die components and sheet metal parts within the model and check for collisions and interference.
- Detect and analyze the collision at each angle of the simulation.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**Run Simulation**

Cooling enhancements

What is it?

Cooling Circuits

Classification of cooling fittings is introduced, once an inlet is selected, the circuit will search the intersected channels if they have the same MW_COOLING_CHANNEL_TYPE attribute according to the inlet channel.

New color setting is set to cooling circuit channels.

You can specify if you want to create a concept symbol or create a component directly.

These enhancements let you flexibly add the classified cooling standard parts. Different color can be set to channels in a circuit. Adding parts directly can reduce the time to add component.

Cooling Fittings

Classification of cooling fittings is introduced. You can now specify if you want to create a concept symbol or directly create a component.

These enhancements let you flexibly add the classified cooling standard parts. Adding parts directly can reduce the time to add component.

Direct Channel

You can now create cooling baffle or cooling channel directly by selecting the specific type.

This enhancement helps you to specify the channel type can create a baffle direction.

Define Channel

Classification of cooling fittings is introduced. You can define the channel into the type by reading the register file definition.

You can flexibly define the channel into the classified types.

Where do I find it?

Application	Mold Wizard
Command Finder	**Cooling Circuits** **Cooling Fittings** **Direct Channel** **Define Channel**

Design Ejector Pin

Use the **Design Ejector Pin** command to command to insert standard ejector components in to your mold assembly.

You can:

- Search and select your ejector components from the Reuse Library.
- Search for sleeve regions to install sleeve pins and recommend ejector length automatically.
- Specify points for positioning ejector positions.
- Edit X and Y ejector pin position values.
- Quickly edit and remove ejectors.
- Assign symbols to ejector pins.

Where do I find it?

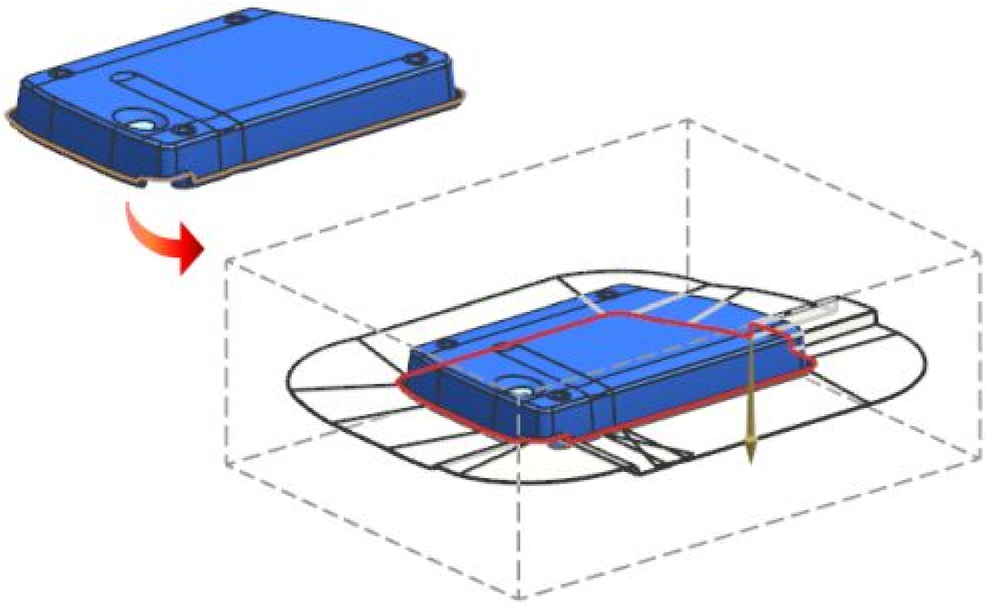
Application	Mold Wizard
Command Finder	**Design Ejector Pin**

Design Parting Surface enhancement

What is it?

You can now use surface extensions to design the parting surface. The surface extensions can be created using connected curves or edges of solid bodies or sheet bodies. To do this, use the **Guided Extension** option.

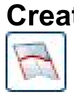
In the example, a parting surface for the core region is designed by creating a surface extension using the connected curves shown.



Why should I use it?

Use the **Guided Extension** method when you want to create a parting surface without additional trimming, or in circumstances that the other methods, such as **Extrude** or **Bounded Plane** cannot create parting surfaces successfully.

Where do I find it?

Application	Mold Wizard
Command Finder	**Design Parting Surface** 
Location in dialog box	**Create Parting Surface** group→**Method** subgroup→**Guided Extension** 

Progressive Die Wizard

Standard parts reuse enhancement

What is it?

You can now browse and select standard parts in the **Reuse Library** when you use the **Standard Parts Library** command.

In previous releases, the standard parts were displayed within these dialog boxes which made the dialog boxes unwieldy. Now these dialog boxes are smaller and take up less design space as the standard parts are displayed in the **Reuse Library**.

To open the standard parts in the **Reuse Library**, you can click **Select Item** in the **Standard Part Management** dialog box. After you select a part from the **Reuse Library**, its parameters are populated in the dialog box. You can change these parameters before you place the standard part in your design.

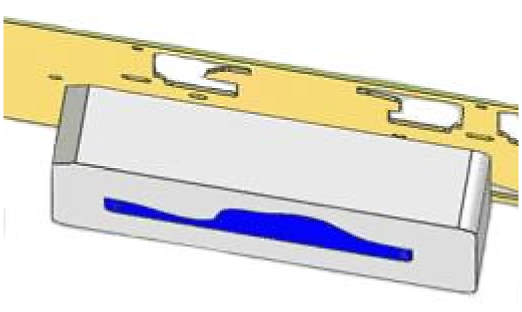
Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Standard Parts**
Location in dialog box	**Standard Part Management** dialog box→**Select Item**

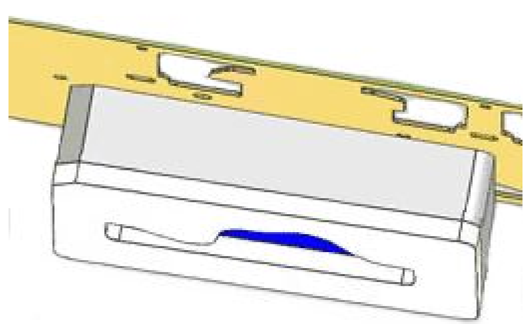
Shim

What is it?

Use the **Shim** command to extrude a shim body from the selected face of a piercing die insert or punch.



No Shim



Shim added

Why should I use it?

The punch and die insert wear out with time. Use this command to add a shim at the top or bottom of the insert or punch in order to maintain the correct height.

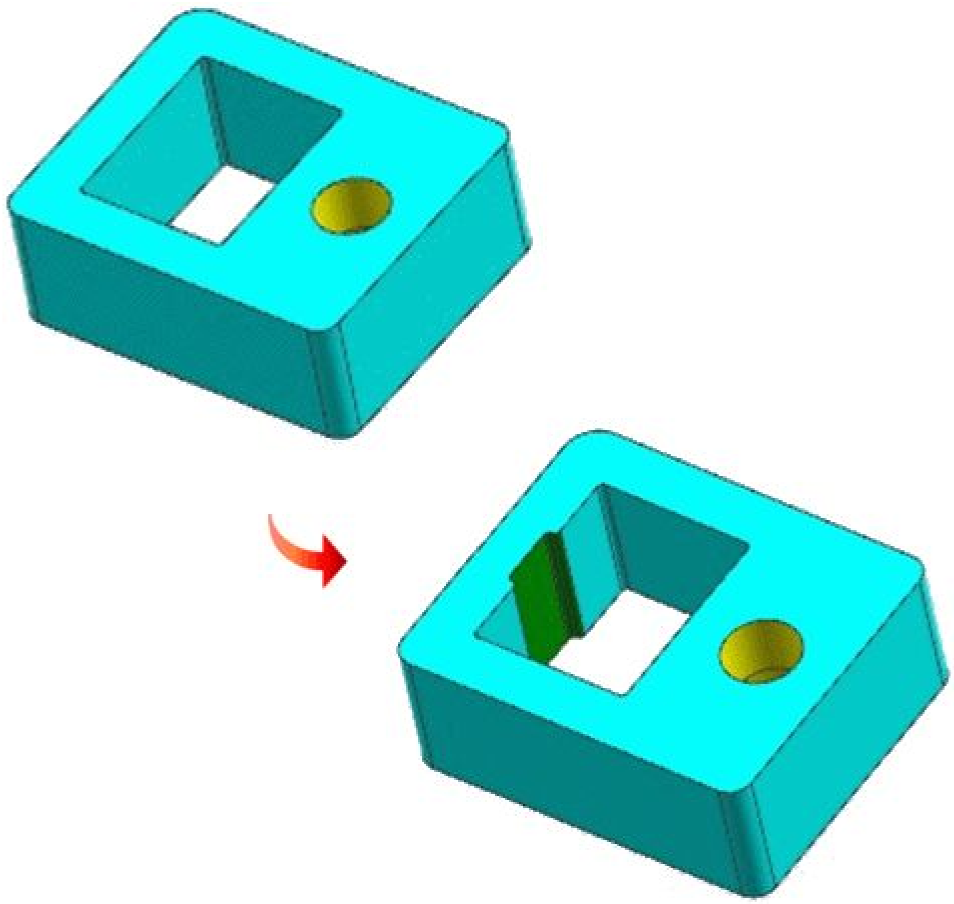
Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Shim**

Stripper Venting

What is it?

Use the **Stripper Venting** command to cut a venting hole in the stripper plate.



Why should I use it?

You can use this command to cut a venting hole in the stripper plate during high-speed stamping.

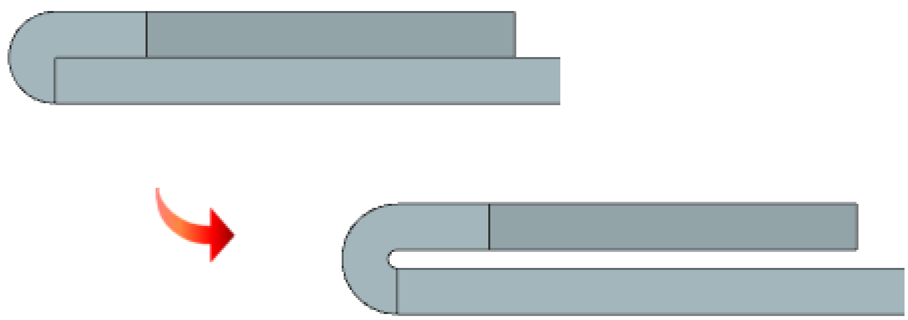
Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Stripper Venting**

Hem Fixer

What is it?

Use the **Hem Fixer** command to add a gap between the base body and the hem flange.



Why should I use it?

You cannot unform a flange if there is no gap between the flange and base sheet body. Use this command to add a gap between them.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Hem Fixer** 

Unfolding Simulation

What is it?

Use the **Unfolding Simulation** command to simulate unfolding of a bend face.

Why should I use it?

You can use this command to check the interference of bend faces with other components during unfolding process.

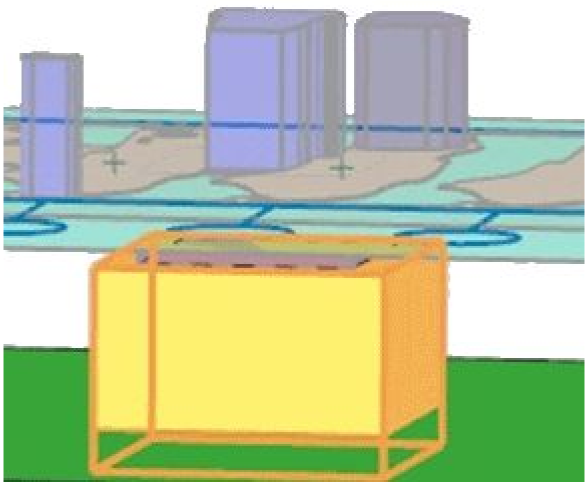
Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Unfolding Simulation** 

Backing Pad

What is it?

Use the **Backing Pad** command to design backing pads for punch and die inserts in casting dies.



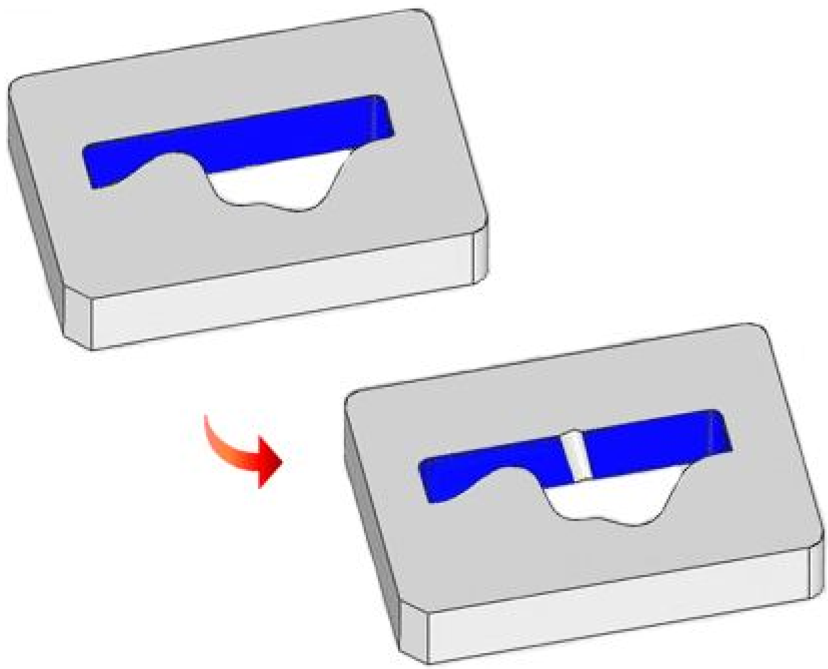
Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Backing Pad** 

Slug Retention

What is it?

Use the **Slug Retention** command to design a slug retention in die openings.



Why should I use it?

Slugs that become dislocated in stamping operations can damage either the stamped part or the die. The slug retention hole helps to keep the slugs from coming out of the die openings.

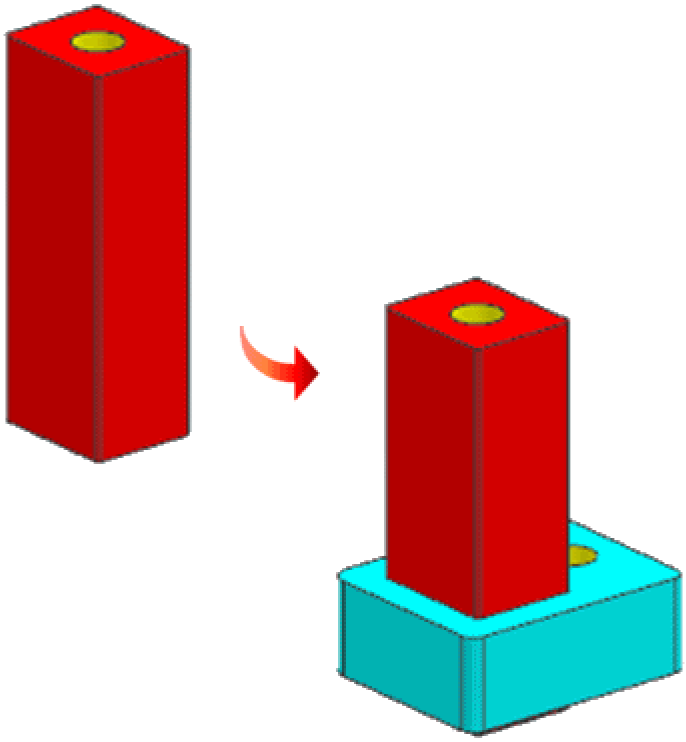
Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Slug Retention**

General Insert

What is it?

Use the **General Insert** command to create outside-insert for punch or die insert.



Where do I find it?

Application	Progressive Die Wizard
Command Finder	**General Insert**

Concurrent Design enhancement

What is it?

You can now:

- Add a task description as an attribute in the task component.
- Save all the master model components, if you are the task owner.

Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard
Command Finder	**Concurrent Design Management**

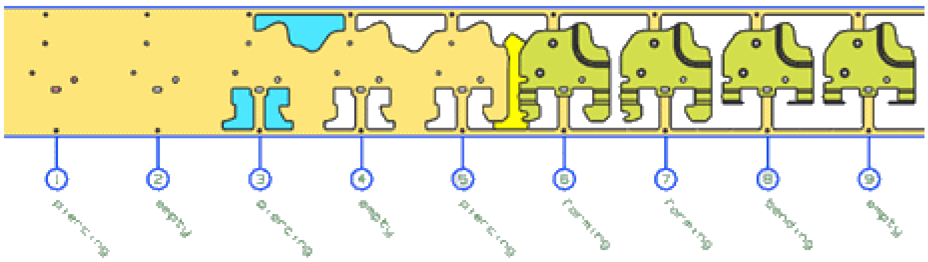
Strip Layout enhancement

What is it?

You can now add a description of the process that is performed on each station.

To add a description, right-click a station name in the **Strip Layout Navigator** and choose **Add Process Description**.

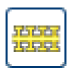
NX also displays the station number inside the circle of each station.



Why should I use it?

You can provide additional information about the process that is performed on any station.

Where do I find it?

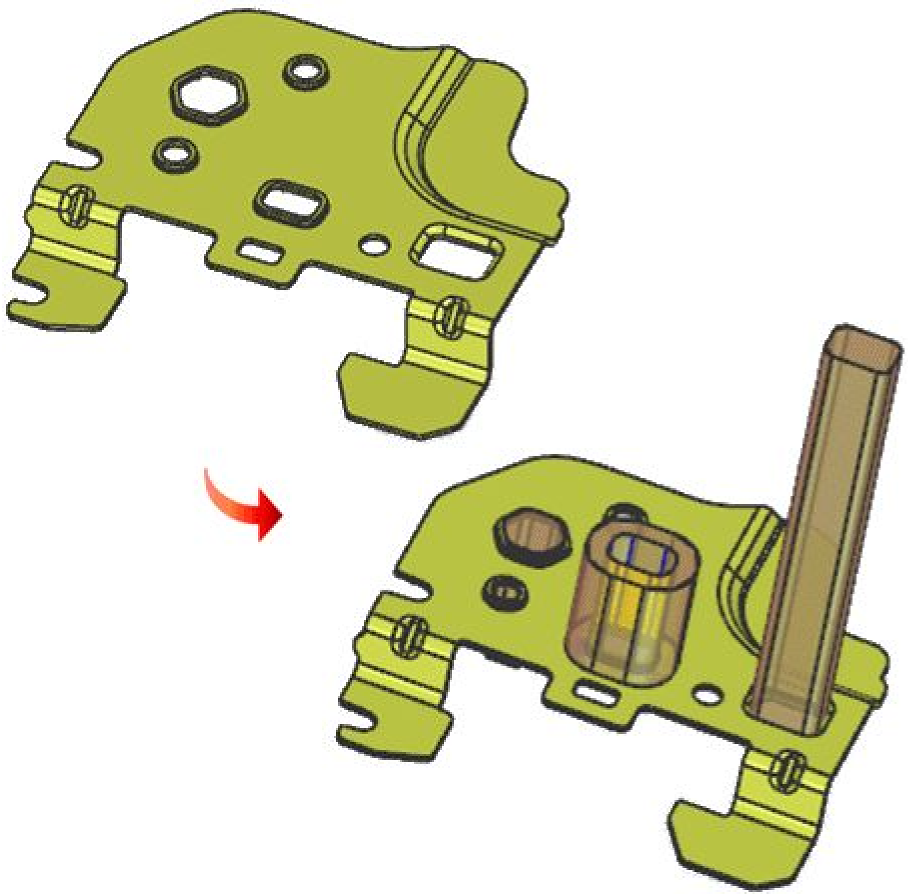
Application	Progressive Die Wizard and Engineering Die Wizard
Command Finder	**Strip Layout** 

Burring Insert Design enhancement

What is it?

You can now design:

- A punch and die to create non-circular burring holes in the product.
- A punch for burring holes without defining a blank.



Why should I use it?

Non-circular burring holes are very common in electronic parts. Use this enhancement to generate such burring holes.

Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard
Command Finder	**Burring Insert Design** 
Location in dialog box	**Burring Insert Design** dialog box→**Burring Insert** group→**Circular or Non-circular** **Burring Insert Design** dialog box→**Burring Insert** group→**Create without blank**

Universal Unform enhancements

What is it?

You can now define a Bend Allowance Formula (BAF) in a spreadsheet. NX uses the user defined BAF to calculate the developed length of bend faces.

Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard
Command Finder	**Universal Unform**
Location in dialog box	**Universal Unform** dialog box→**Define Bend** group→**Developed Length Calculation**

Direct Unfolding enhancements

What is it?

You can now define a Bend Allowance Formula (BFA) in a spreadsheet. NX uses the user defined BAF from the spreadsheet to calculate the developed length of bend faces.

Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard
Command Finder	**Direct Unfolding**
Location in dialog box	**Direct Unfolding** dialog box→**Define Neutral Factor** group→**Developed Length Calculation**

Prebend enhancement

What is it?

When you create a prebend, NX now combines the two **Extrude** and **Divide Face** features into a single **Prebend** feature in the **Part Navigator**.

Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard
Command Finder	**Direct Unfolding** **Bend Operation**

Component Drawing enhancement

What is it?

You can now pass the selected part attributes to the Master Model drawing part.

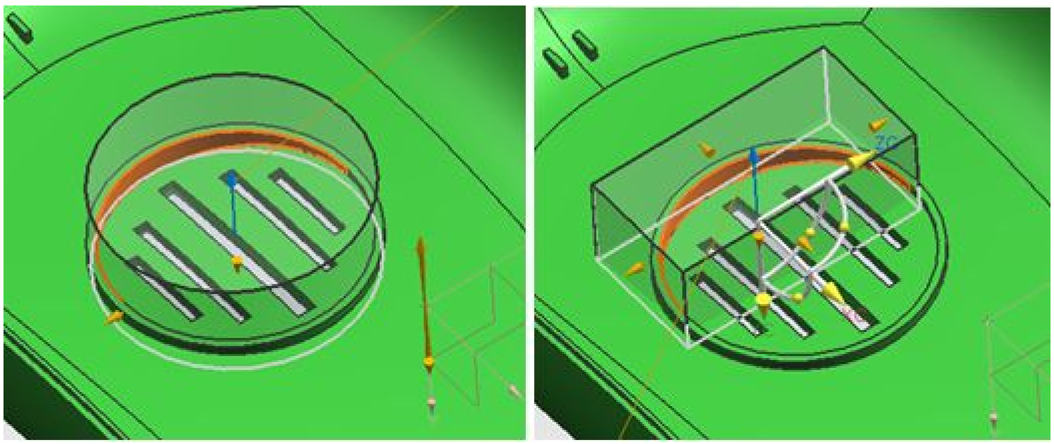
Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard

Command Finder	**Component Drawing**

Create Box

Use this command to create a block or cylinder which envelopes the selected objects. This command replaces the previous box feature based on Knowledge Fusion.



You should use this command when you want to create a block or a cylinder which envelopes the contour with selected faces, edges, curves, points, solid bodies or facet bodies.

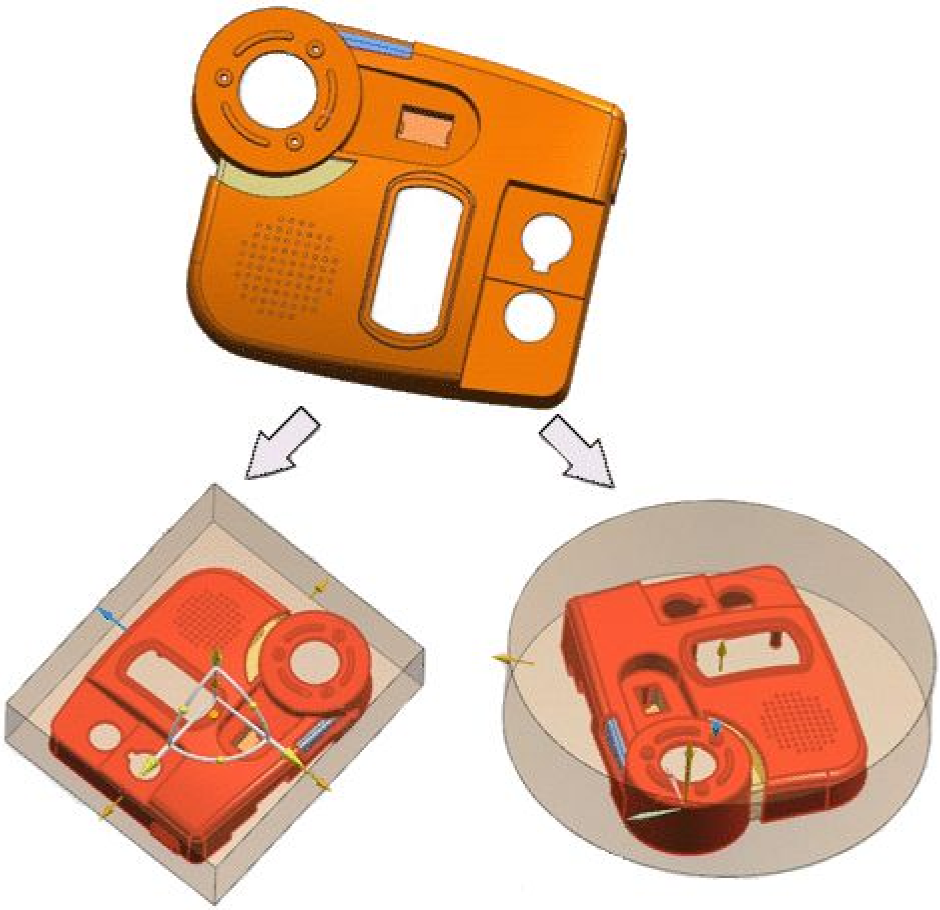
Where do I find it?

Application	Mold Wizard, Progressive Die Wizard, and Electrode Design
Command Finder	**Create Box**

Stock Size enhancements

What is it?

Stock Size is enhanced with a smart object in NX. It now allows being modified with six clearances in six directions. It also provides an option to set zero a clearance value automatically if there are extreme faces in selected body in the specified direction.



Why should I use it?

You should use it when you want to calculate the stock (block or cylinder) size which envelopes the contour with selected solid bodies.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**Stock Size**

Motion Simulation commands

What is it?

The **Tooling Motion Simulation** command is removed.

Use the following commands to set up the kinematic model, define the cam, define the lifter, define user defined motion, and run the simulation.

Motion Preprocess

Use this command to set and load the kinematic model, and mount the sheet metal or plastic parts of the die or the mold on the kinematic model.

Why should I use it?

You can use this command to prepare your motion simulation data.

You can:

- Clone a predefined kinematic model for tooling into your current assembly or a directory you specify.
- Change the kinematic model. NX displays a message window to confirm whether you want to remove the current kinematic model and add another one. If you click **Yes**, you must mount the components again and redefine the cams.
- Generate control data and import it to a kinematic model according to the die settings. You can also change to different control data and re-read the control data.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**Motion Preprocess**

Define Cam

Use this command to create, edit, or delete a linear cam, a rotary cam, a rocker cam, a user-defined cam, or cushion programming. You can select solid bodies, sheet bodies, or faceted bodies that have solid or sheet bodies associated with it, and use them to define the cam.

Why should I use it?

A predefined kinematic model contains kinematic data about basic cam actions. You can use this command to define the kinematic data for various types of cams. NX automatically calculates and stores the cam data in the kinematic model.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**Define Cam**

Define Lifter

Use this command to create, edit, or delete a generic lifter or a lifter on a lifter. You can select solid bodies, sheet bodies, or faceted bodies that have solid or sheet bodies associated with it, and use them to define the lifter.

Why should I use it?

A predefined kinematic model contains kinematic data about basic lifter actions. You can use this command to define the kinematic data for various types of lifter actions. NX automatically calculates and stores the lifter data in the kinematic model.

Where do I find it?

Application	Progressive Die Wizard and Mold Wizard
Command Finder	**Define Lifter**

User Defined Motion

Use this command to add user defined motion data to your kinematic model.

Why should I use it?

You can define:

- Linear movement along a specific vector.
- Angular movement about a specific axis.

The motion curve is a set of 360 values, one for each timing angle 0-359, that determines how far to move along or about the axis at each point of the simulation. You can use the simple motion curves created by NX, or supply your own values using a comma separated values (CSV) file.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**User Defined Motion**

Run Simulation

In Progressive Die Wizard, use this command to view an animation that shows the kinematics of all the automatic and user-identified components in the model.

Why should I use it?

You can:

- View the die components and sheet metal parts within the model and check for collisions and interference.
- Detect and analyze the collision at each angle of the simulation.

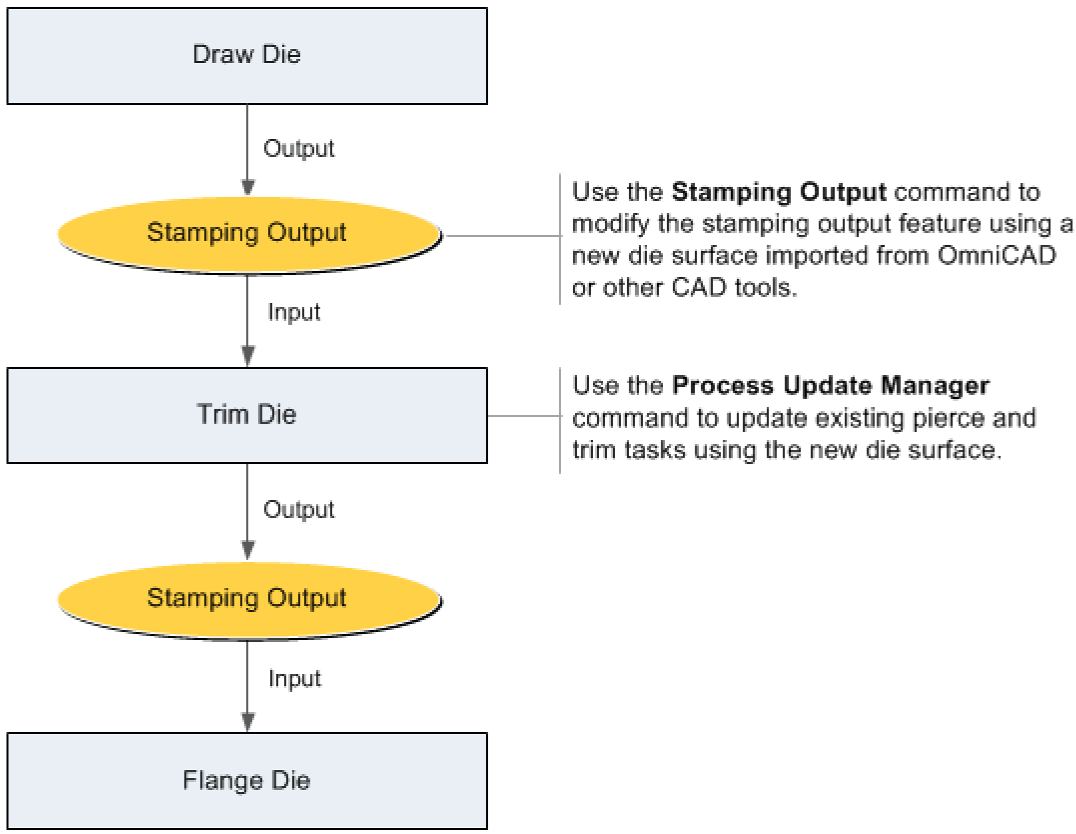
Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**Run Simulation**

Die Engineering

Using a new die surface in an existing stamping die workflow

This workflow shows how to use a new or modified die surface in an existing stamping die workflow.



Addendum Section enhancements

What is it?

You can improve the quality of the addendum section by specifying a point on the intersecting curve.

NX uses the specified point as a reference point and displays a temporary point on the addendum section. You can use this temporary point to verify the quality of the addendum section.

You can also measure the distance between the start point of the addendum section curve and the trim point. To do this, use the new **Measure Distance** button on the **General** tab in the **Section Parameters** dialog box.

Where do I find it?

Application	Die Engineering and Engineering Die Wizard
Prerequisite	You must be in the Modeling application.
Command Finder	**Addendum Section**
Graphics window	Right-click an addendum section feature→**Edit Addendum Section**
Location in dialog box	**Section Parameters** dialog box→**General** tab→**Measure Distance**

Develop Trim Line enhancements

What is it?

You can generate the trim line profile using the FEM method that uses the One-Step solver.

When you use this command, you can also:

- Select sheet material either from the NX material library or local material library.
- Define density, thickness, Poisson's ratio, Yield stress, and so on.
- Specify solver parameters such as convergence level, iteration steps, and so on to increase the accuracy of the trim line profile.

Where do I find it?

Application	Die Engineering and Engineering Die Wizard
Prerequisite	You must be in the Modeling application.
Command Finder	**Develop Trim Line** 

Stamping Output enhancements

What is it?

You can update a die surface in OmniCAD or another CAD tool, import the surface into NX, and use it in the next stamping operation without changing the existing stamping die workflow.

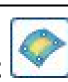
Use the new **Stamping Output** command to modify or create a stamping output feature that uses the imported die surface. The stamping output feature includes the legacy surface and the new die surface.

To use the new die surface in the next stamping operation, you must update existing dependent features such as pierce and trim tasks that use the legacy die surface. Use the **Process Update Manager** command to manage the existing pierce and trim tasks.

Why should I use it?

To meet manufacturing requirements, you can use an updated die surface from another CAD tool in an existing stamping die workflow. This capability helps to reduce design time and increase productivity.

Where do I find it?

Application	Die Engineering
Prerequisite	You must be in the Modeling application.
Command Finder	**Stamping Output** 

Process Update Manager

Use the **Process Update Manager** command to display and update existing pierce and trim tasks after you modify the stamping output feature.

In a stamping die workflow, when you modify the stamping output feature using a new die face, you must update the existing pierce and trim tasks that use the old die face.

When you use the **Process Update Manager** command, you can:

- Select the type of task that you want to update. Depending on the selected pierce or trim task type, NX maps the existing tasks to the new die face and displays a list in the dialog box. The list includes the name of the task, the stamping operation associated with the task, and the status of the task.
- Update the existing tasks by selecting each task from the list. NX changes the status of the updated tasks to **Completed**.
- View and compare the old and new die faces in the graphics window by setting the display status of the die faces.
- View existing tasks for a single operation or all stamping operations.

Where do I find it?

Application	Die Engineering
Prerequisite	You must be in the Modeling application.
Command Finder	**Process Update Manager**

Stamping Carryover enhancements

What is it?

You can copy a stamping output feature that includes existing and new die surfaces, from the previous stamping operation to the current stamping operation.

This helps you to perform die engineering tasks on a new die surface and quickly complete the new design workflow.

Where do I find it?

Application	Die Engineering
Prerequisite	You must be in the Modeling application.
Command Finder	**Stamping Carryover**

OmniCAD for NX

OmniCAD commands enhancements

What is it?

You can morph objects in the NX CAD environment using the following integrated OmniCAD commands. When you use these commands, NX now creates features for each of these commands.

OmniFree Transformer Use this command to morph objects using points and curves as constraints.

OmniMesh Transformer Use this command to morph objects using CAE Meshes, STL files, or point clouds.

To edit a feature, double-click it in the **Part Navigator**.

Where do I find it?

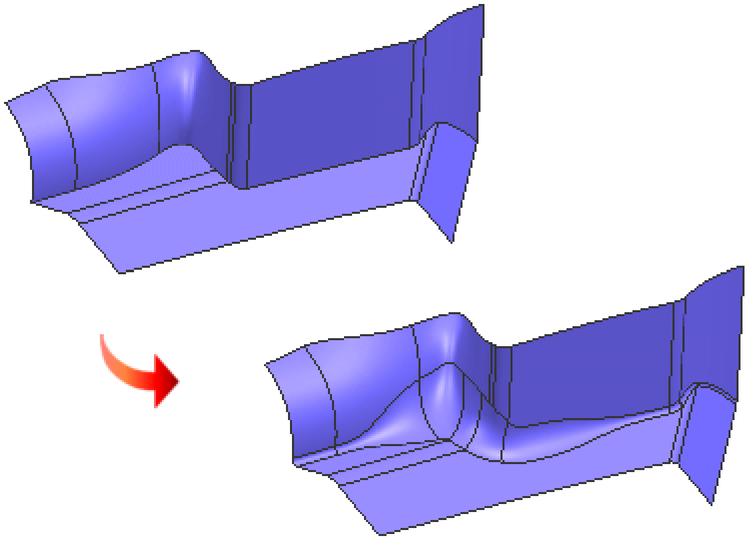
Application	OmniCAD for NX
Prerequisite	To use OmniCAD commands, you must run the `InstallSentinel.bat` file which is available in the *ugii/omnicad* directory.
Command Finder	**OmniFree Transformer** , **OmniMesh Transformer** 

Flow Blend (10.0.1)

Use the **Flow Blend** command to create constant or variable blends along complex faces.

You can:

- Blend complex faces that have a small curvature.
- Patch the blended faces to the original body.
- Smoothen variable blends.



Where do I find it?

Application	OmniCAD for NX
Prerequisite	To use OmniCAD commands, you must run the `InstallSentinel.bat` file which is available in the *ugii/omnicad* directory.
Command Finder	**Flow Blend** 

Weld Assistant

Fillet Weld enhancements

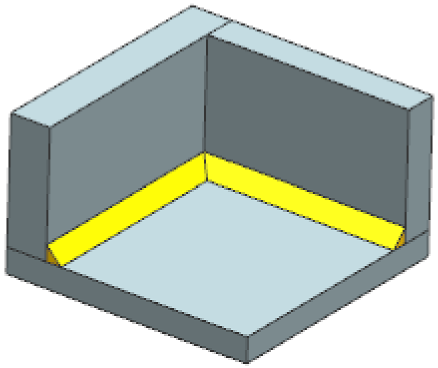
What is it?

The **Fillet Weld** command has the following workflow and selection enhancements.

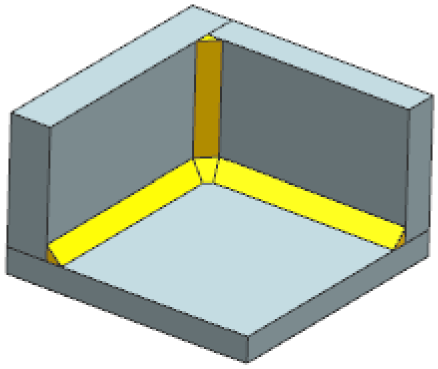
Workflow enhancements

When you create a fillet weld, you can now:

- Create a single weld that goes around an inside corner.



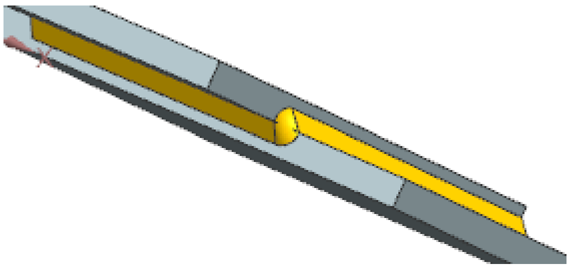
You can also select the face of an existing fillet weld as a face in the face set.



- Create a continuous weld.

  To do this, use the **Roll Around** option. Continuous welds let you complete the welding in a single operation.

  The example shows a continuous weld which bridges the corners and rolls from the top of a plate to the side of the plate.



- Create non-associative fillet welds.

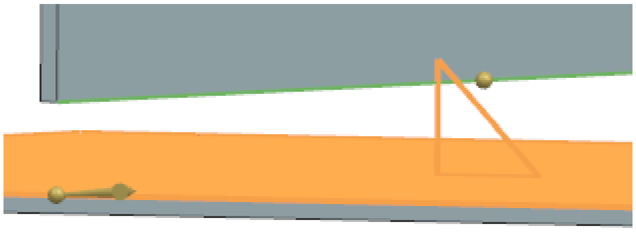
  To do this, clear the **Associative** check box. When fillet welds are non-associative, NX does not automatically update them when the WAVE linked geometry changes.

Selection enhancements

When you create a fillet weld, you can now:

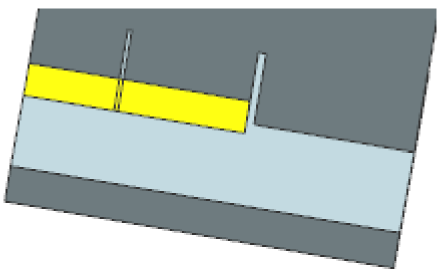
- Let NX automatically select the edge sets of the bodies to be welded when a gap exists between two face sets.

  To do this, use the **Extend Edges** option. This option helps you to avoid errors in manual selection.

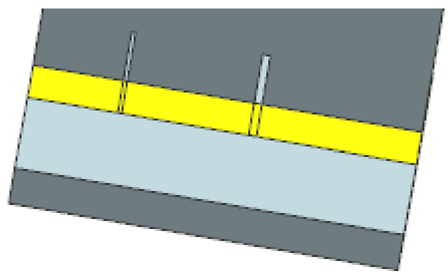


- Specify the distance over which the fillet weld can continue if a small gap exists in the face.

  To do this, use the **Face Fill Gap** option. This option helps you to eliminate the selection of multiple edges or the creation of additional fillet welds.



Face Fill Gap = 2



Face Fill Gap = 3

To change the default distance value, use the **Face Fill Gap Distance** customer default.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

Where do I find it?

Application	Modeling
Command Finder	**Fillet Weld** 

Location in dialog box	**Face Sets** group: **Extended Edges** **Construction Method→Roll Around** **Settings** group: **Face Fill Gap** **Associative** ☐

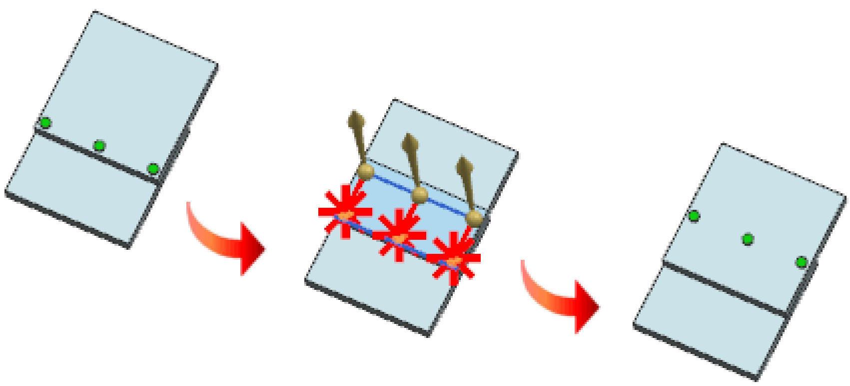
Weld Point Wizard

What is it?

The **Weld Point Wizard** replaces the **Weld Point** dialog box and provides additional functionality.

Moving weld points

You can move an existing weld point to a new location, and maintain its ID and attributes. To do this, use the **Select Reuse Features** check box.



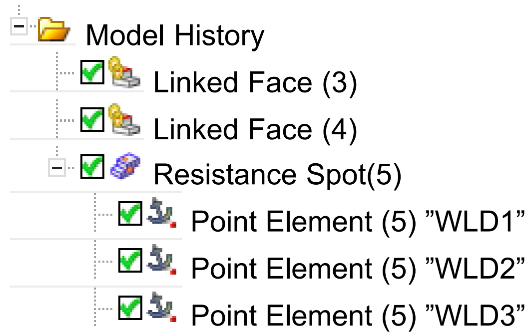
If the weld point reuse feature is not mapped as desired, you can also specify a weld point feature from which you want to map. To do this, use the **Select Mapping Feature** option.

Creating single weld point features

You can create single weld point features without the weld feature set. To do this, use the **Create Single Features** check box.

Create Single Features =

Create Single Features =



When you create weld point features, NX:

- Maintains connected part information for each weld through the connected part occurrences instead of through the linked faces. This reduces the need to use the **Connected Face Finder** command.

  A weld point does not go into a retained state if the link to a parent face is broken.

- Displays weld point locations before you generate the weld points.

- Infers creating geometry of each weld point without a feature set.

Automatic updates and editing welds

You can automatically update weld locations by associating a weld point with its creation data. To do this, use the **Automatic Update** check box.

You can also edit multiple welds regardless of how they were created.

Why should I use it?

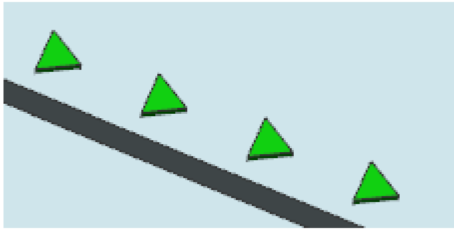
When you use the wizard, you have greater flexibility when you create weld point features, and better visual feedback of these features. You can also create multiple weld points simultaneously without creating a feature set.

Where do I find it?

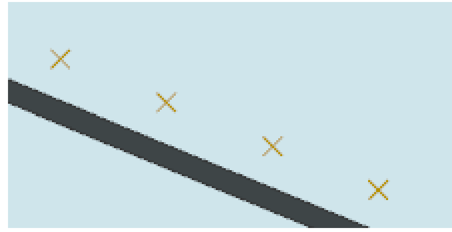
Application	Modeling
Command Finder	**Weld Point Wizard**

Solid Weld Point Display

Use the **Solid Weld Point Display** option to display a weld point as a solid or a point.



Solid Weld Point Display turned on



Solid Weld Point Display turned off

To change the shape and dimensions of the weld point display, use the **Point Locator** customer defaults. You can also specify unique colors for two panel, three panel, and four panel welds.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default**.

Where do I find it?

Application	Modeling
Command Finder	**Solid Weld Point Display**

Converting legacy welds

Use the **Convert Legacy Weld Points** weld utility to convert pre-NX 10 weld point features to the same style as the weld point features that are created by the **Weld Point Wizard**.

When you run the **Convert Legacy Weld Points** weld utility, NX:

- Converts single weld point features to an NX 10 single weld point feature. NX does not create a grouping feature.
- Deletes the pre-NX 10 feature set and any hidden linked faces. NX creates a reference to the connecting component to retain the connected part information.
- Maintains associativity of the mirrored and translated weld point features.
- Uses the **Legacy Welds (pre NX10)** customer defaults to set the size of the custom weld points that are generated as solids, and maintains the color.
- Uses the **Point Locator** customer defaults for weld points that are generated without a solid.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default**.

Note

You cannot use this utility to convert pre-NX 5 weld points. To convert the pre-NX 5 weld points, you can do the following:

1. Export the locations of the weld points to a CSV file using the **Export CSV** command.

2. Import the same CSV file using the **Import CSV** command.

Where do I find it?

Application	Modeling
Prerequisite	Set the **Point Locator** customer defaults similar to the **Legacy Welds (pre NX10)** customer defaults.
Command Finder	**Weld Utilities**
Location in dialog box	**Convert Legacy Weld Points** check box

Converting legacy welds (10.0.1)

When you run the **Convert Legacy Weld Points** weld utility, you can specify if you want to simultaneously convert all or selected pre-NX 10 weld point features. To do this, use the **All in Work Part** option or the **Selected Only** option.

Where do I find it?

Application	Modeling
Prerequisite	Set the **Point Locator** customer defaults similar to the **Legacy Welds (pre NX10)** customer defaults. To select the **Selection Scope** options, you must select the **Convert** check box.
Command Finder	**Weld Utilities**
Location in dialog box	**Convert Legacy Weld Points** group→**Convert** check box **Selection Scope** subgroup→**All in Work Part** or **Selected Only**

Weld Advisor enhancements

What is it?

The Weld Advisor command provides additional functionality.

- If the faces required for a **Weld Advisor** check are not available, NX automatically finds the faces needed for the check.

- NX displays different icons for faces based on whether **Connected Face Finder** is used to find the faces or if existing faces are used in the **Weld Advisor** check.

Connected Face Finder faces used in Weld Advisor check	Existing faces used in Weld Advisor check
Connected Face Sets Connected Face Set 1 Connected Face Set 2	Connected Face Sets Connected Face Set 1 Connected Face Set 2

Why should I use it?

These enhancements help you to save design validation time.

Example

If you import weld points using the **Import CSV** command, you need not use the **Connected Face Finder** command to find the faces required for the **Weld Advisor** check.

Where do I find it?

Application	Modeling
Command Finder	**Weld Advisor**

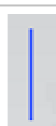
Joint Mark enhancements

What is it?

NX now provides the following functionality.

Selecting custom points

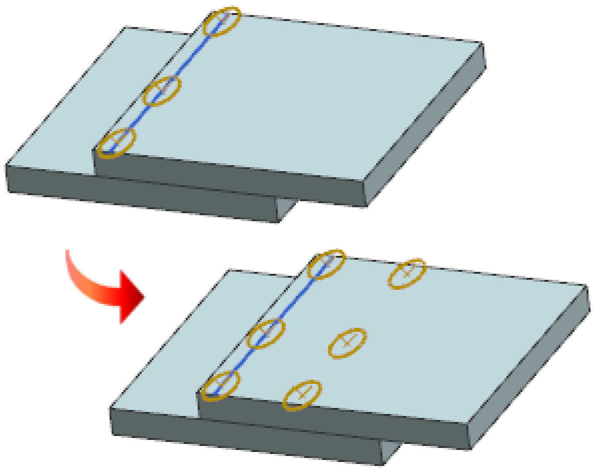
You can select from any of the following custom symbols to represent the joint mark.

	Ellipse
	Half circle
	Line

In previous releases, you had to create custom symbols, and configure NX to display the custom symbols in the **Reuse Library**.

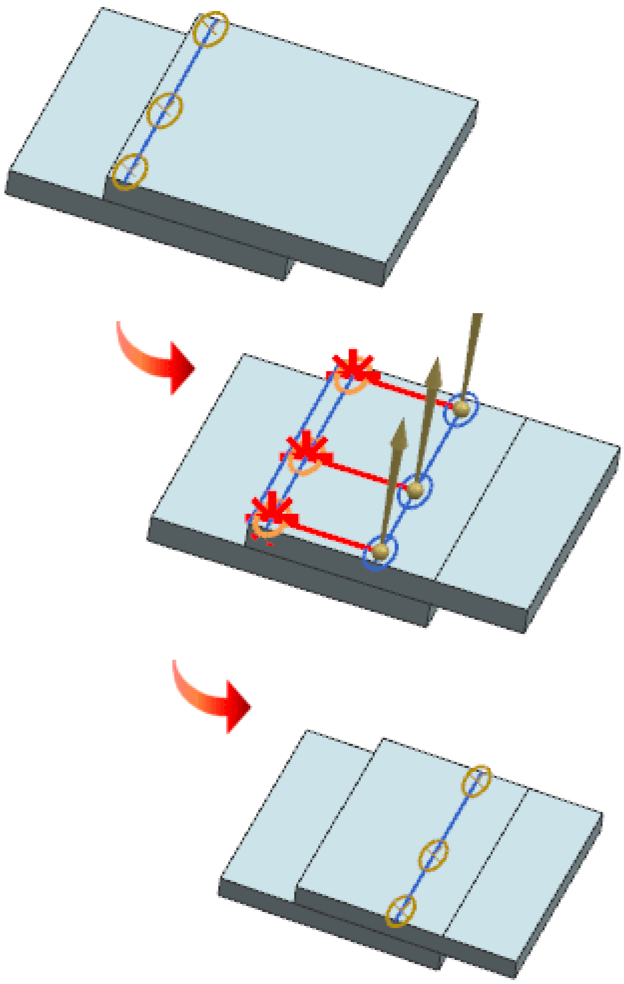
Creating joint marks through translation

You can create joint marks by translating existing joint marks. To do this, use the **Translate** option.

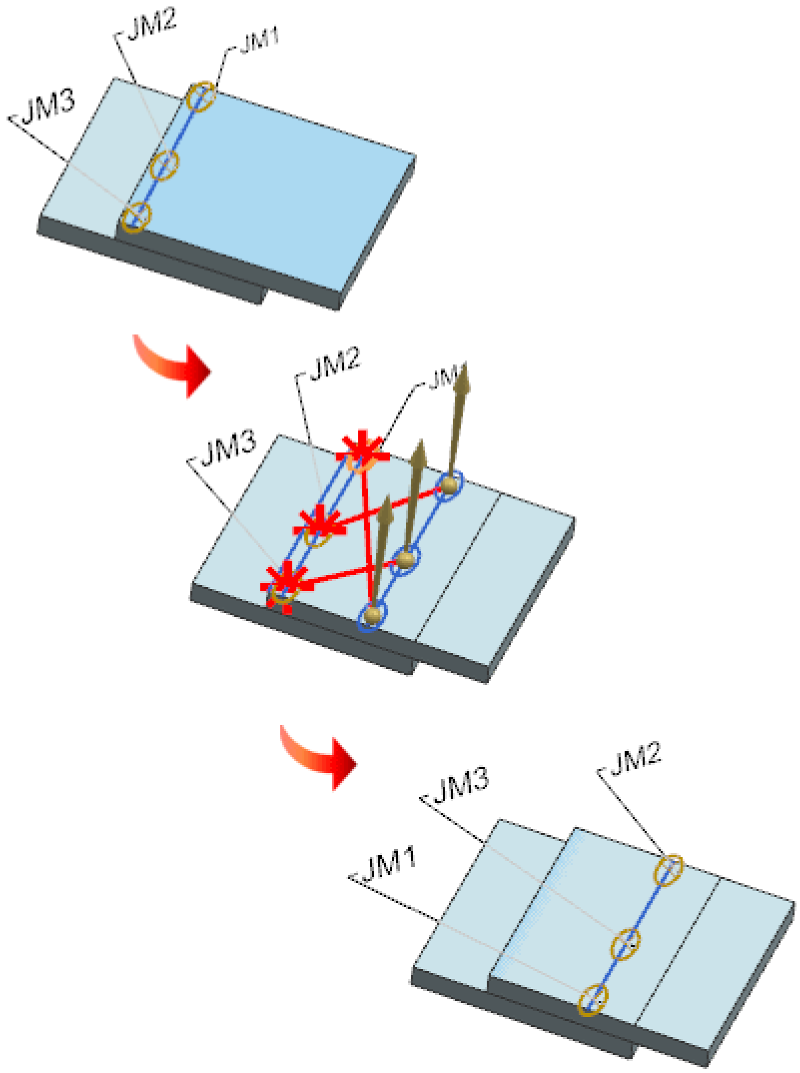


Moving joint marks

You can move an existing joint mark to a new location, and maintain its ID and attributes. To do this, use the **Select Reuse Feature** option.

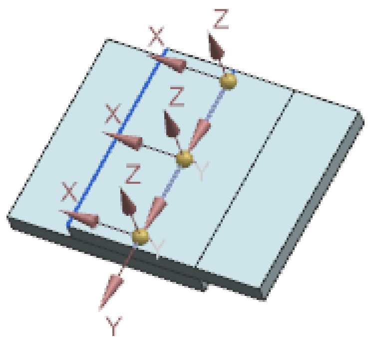


If the joint mark reuse feature is not mapped as desired, you can specify a joint mark to map the feature from when you move a joint mark. To do this, use the **Select Mapping Feature** option.

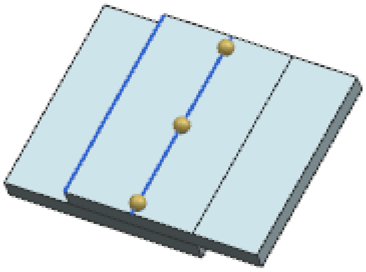


Displaying the coordinate system

You can display the coordinate system when you create joint marks. To do this, use the **Show Coordinate System** ☑ check box.



Show Coordinate System = ☑



Show Coordinate System =

Where do I find it?

Application	Modeling
Command Finder	**Joint Mark**
Location in dialog box	**Method** step→**Method** group→**Method** list→**Translate** **Review/Edit** step→**Redefine Locations** group→**Select Reuse Features**, **Select Mapping Feature** **Settings** step→**Settings** group→**Show Coordinate System** check box

Structure Welding

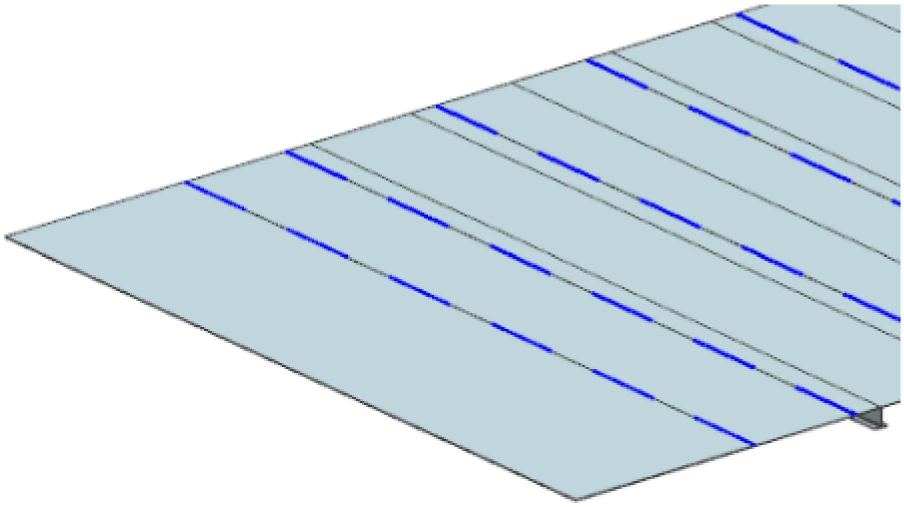
Skip welding joint

What is it?

When you use the **Welding Joint** command, you can now create welding joints that are split into segments that have skips or spaces between them. These joints are called *skip joints*. You can specify the length for the segments and the spacing distance between the segments.

Example

The example shows skip welding joints used to weld multiple plates. Each weld segment has a length of 1500 mm, and is spaced at 1000 mm from the previous segment.



Where do I find it?

Application	Modeling
Command Finder	**Welding Joint**
Location in dialog box	**Welding Joint**→**Create**/**Modify** step→**Split Joints** group→**Split joints** list→**Skip**

Inheriting welding characteristics

What is it?

You can specify whether a welding joint inherits welding characteristics from the parent welded plate or pipe when you:

- Weld plates designed in Ship Design.
- Weld pipes designed in Routing.

You can also change the value of a selected characteristic. To do this, double-click the cell in the **Inherit** column of the characteristic's row, and update the value.

Where do I find it?

Application	Modeling and Routing
Command Finder	(Modeling) **Welding Joint** (Routing) **Pipe Welding Joint**
Location in dialog box	**Welding Joint** or **Pipe Welding Joint**→**Create**/**Modify** step→**Welding Characteristics** group

Weld attributes in 4GD

When you create a weld design feature in 4GD, you can add custom attributes to the feature from Teamcenter using Attribute Groups and Managed Attribute Groups.

To do this, you must set the following **Welding Characteristics** customer default:

HasAttributeGroup = TRUE

> **Tip**
>
> To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

If you change weld attributes in Teamcenter, you must load the part fully to synchronize the changes to the weld design feature.

Chapter 8: Data translation

NX to JT

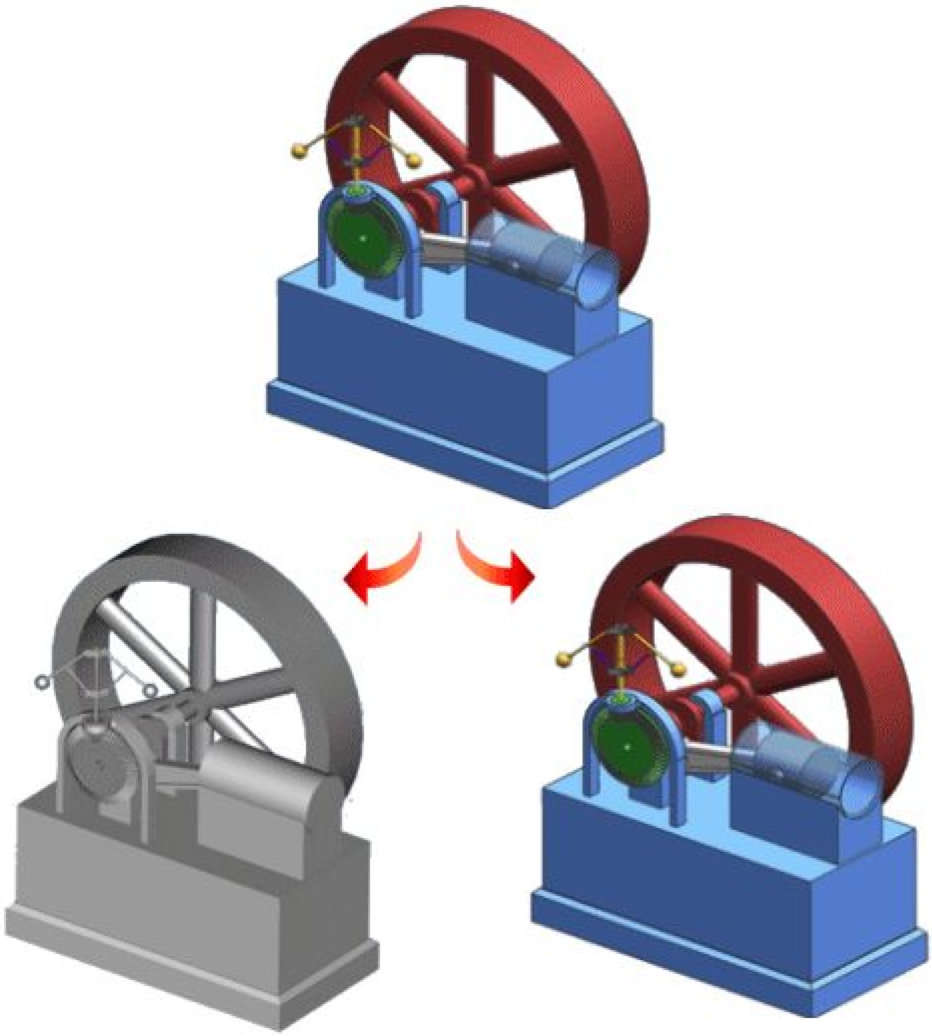
Support for assembly level color and translucency overrides in JT files

What is it?

When you export an NX assembly to a JT file, NX now exports the color and translucency overrides applied on the components and bodies inside the component of an assembly to the assembly JT file.

When you view the assembly JT file in a JT viewer such as Teamcenter Visualization, you can see the correct representation of an assembly in terms of color and translucency applied on the components or bodies inside the components at the assembly level.

The following example shows the display difference between an assembly exported to a JT file in versions earlier to NX 10 and from NX 10.



Pre-NX10 NX10

Note

Color or translucency overrides applied on individual faces of bodies inside components are not preserved in the assembly JT file.

Where do I find it?

Command Finder	**Export JT File**
Command line	Command line application **ugtopv** invoked with the `-honour_structure` option

Support for NX lightweight representation in JT files

What is it?

You can now export an NX lightweight representation, also known as a faceted representation, to a JT file, as Level of Details (LODs) in the tessellated object. To do this, use the **Use NX Lightweight Representation** option.

Why should I use it?

Because the NX to JT translator does not have to re-tesselate objects when you export lightweight representations, processing time and memory requirements are improved.

Where do I find it?

Command Finder	**Export JT File**
Location in the dialog box	**Export JT** dialog box→**Tessellation** tab→**Level of Details** group→**Use NX Lightweight Representation**

Support for ESKD standard entities in NX translators

What is it?

When you export data using the 2D Exchange Translator or the DXF/DWG translator, NX exports the following ESKD data:

- ESKD symbols
- ESKD section lines
- Viewing direction reference arrows for projected views
- Dimensions with text above the stub style
- Limits and fits tolerance types

Where do I find it?

Command Finder	**Export AutoCAD DXF/DWG File** **Export 2D Exchange File**

2D Exchange translator and AutoCAD DXF/DWG translator enhancements

What is it?

When you use the 2D Exchange translator to export NX data to a 2D part file or use the AutoCAD DXF/DWG translator to export data to the 2D format, the translator exports:

- Wireframe edges with face colors.
- Pictorial and half-pictorial section lines.
- Newly added NX Drafting and GD&T symbols such as Independency, Continuous Feature, Statistical, Orientation Constraint, Datum Translation, Fit Function, and Safety Compliance.
- Copyright, Registered Trademark, Farad and Centerline symbols added in NX using block font characters.

The 2D Exchange translator also adds the following information about dimension translation to the 2D Exchange log file.

- Summary of dimensions available in input part.
- Summary of dimensions exported to the 2D part.
- Dimensions exported as grouped data.

  The information about dimensions exported as grouped contains dimension type, measurement values, the associated view of the dimension, the name of the drawing Sheet on which the dimension exists, and the reason why the dimension is converted to grouped data.

Where do I find it?

Command Finder	**Export 2D Exchange File** **Export AutoCAD DXF/DWG File**

Exporting center lines and center marks

AutoCAD DXF/DWG translator

When you export NX data that contains centerlines to DXF/DWG file, you can now create a customized centerline font to match its display properties in NX. To do this, set the following keyword in the settings file.

```
EXPORT_CENTERLINE_AS=CUSTOM
```

To get the exact display, you can also export the centerlines as block reference to DXF file. To do this, set the following keyword in the settings file.

```
EXPORT_CENTERLINE_AS=BLOCK
```

You can use this option to export centerline as lines, arcs and/or splines with same display as that of NX. A centerline font consists of a unique combination of dashes, gaps and long-dashes.

By default, the settings file does not contain this option. You must add this option to the settings file, or else the translator exports the centerlines with the CENTER font to the DXF file

2D Exchange translator

You can now export linear 3D Centerline, linear PMI Centerline, and PMI Center Marks as real in the 2D part. The exported centerlines retain the properties such as display, dimension, and association.

When you export drawings to the 2D part file, the translator exports:

- Linear 3D and PMI centerlines as 2D centerlines.
- PMI Center Marks as Center Marks in the 2D part.

Where do I find it?

Command Finder	**Export AutoCAD DXF/DWG File** **Export 2D Exchange File**

Support for PMI data in NX translators

What is it?

When you export data using the 2D Exchange Translator or the DXF/DWG translator, NX exports the following PMI and additional data:

- PMI annotations
- Specialized PMI
- Security markings
- Section views

Where do I find it?

Command Finder	**Export AutoCAD DXF/DWG File** **Export 2D Exchange File**

Support for international characters in NX data translators

What is it?

NX supports international characters for the following entities:

- Names of files, objects, interface labels, and options in the settings file
- Attribute titles and values
- Messages in the log file

  Note

  Before you export data to CAD tools that do not support international characters, you must replace non-English characters with English characters in the parts, and rename the parts in NX.

Support for multifield keys in NXJT and MCAD translators

What is it?

You can now export multifield key based NX items to other CAD formats. NX generates names for the output and log files using multifield key values.

Multifield keys

Multifield keys are identifiers assigned to each object to ensure their uniqueness in the database. The unique identifier is composed of a domain name and a combination of the object's properties. For example, the default multifield key for item business objects is **Item{item_id}**. For more information, see Teamcenter Help.

Chapter 9: System administration

Internationalization and Localization

Using local language in NX applications

What is it?

You can now enter part names, attributes, parameters, file names, design notes, and so on in all local language in NX applications.

Chapter 10: 4th Generation Design (4GD)

Overriding geometry in 4GD

What is it?

4GD is enhanced in the following ways:

- The **Assembly Cut**, **Promote**, **Define Deformable Part**, and **Deform Component** commands are now available.

- You can create an override part for a reuse design element or subordinate in the context of the collaborative design.

You can use the **Create Override Part** command to create an empty override part. After you create an override, you can choose the shape you want to edit in the **Assembly Navigator**.

Why should I use it?

Use these commands when you want to change the shape of an object in the context of the collaborative design.

Where do I find it?

Command Finder	**Assembly Cut** **Promote** **Define Deformable Part** **Deform Component** **Create Override Part**

Sorting 4GD components in the Assembly Navigator

What is it?

You can now sort components in the **Assembly Navigator** if you are working in a 4GD environment. In a workset, you can sort partitions, subsets, and design elements by alphabetical, alphanumerical, and chronological order. You can also create a user-defined order for design elements and subsets.

Why should I use it?

Sorting 4GD objects in the **Assembly Navigator** can make it easier to find and focus on the objects of most interest. Being able to sort 4GD objects lets you:

- Find design elements (DE) more easily.
- Put the subset of most interest at the top of a list.
- Group related design elements together.
- Find new design elements in a subset.

Where do I find it?

Prerequisite	You must be in a Teamcenter environment and have 4GD installed.
Command Finder	**Reorder Components**
Menu	**Assemblies→Navigator Order→Edit Orders in Part**

Mirror Assemblies Wizard

What is it?

If you are in a Teamcenter environment, you now use the business logic set up for your site when you create new mirror parts using the **Mirror Assembly Wizard** (MAW). The MAW also supports the new Teamcenter multi field key attributes.

The new items created in Teamcenter for any newly created mirror parts will be of the same type as the source parts being mirrored. When you mirror a part or assembly, you create a new part or assembly, not a new revision.

Note

You can no longer change the item type for mirrored parts with the **Mirror Assembly Wizard**.

Why should I use it?

You can rename all the new objects and their attributes, including Teamcenter multi field attributes, at one time in a single mirroring process. You can see and validate all the attributes as you mirror them.

Where do I find it?

Prerequisite	You must be in a Teamcenter environment.
Command Finder	**Mirror Assembly**

Visual Reports

What is it?

There are three new out of the box visual reports for 4GD objects:

- **Design Model Element Has Higher Revision ID**

  Use this report to display the list of Design Model Elements (DE) with higher revision ID that are currently loaded in your session.

- **Design Model Element Source Item Revision Has Higher Revision**

  Use this report to display the list of Design Model Elements (DE) with higher revisions that are currently loaded in your session.

- **Compare Realized Objects in Two Subsets**

  Use this report to display the list of Design Model Elements (DE) that are currently loaded in your session and the names of two loaded subsets for the reporting scope.

Where do I find it?

Prerequisite	You must be in a Teamcenter environment and have 4GD installed.
Command Finder	**Start Visual Reporting**
Resource bar	**HD3D Tools→Visual Reporting**

Modifying a subset using a Teamcenter option set

What is it?

The **Subset Navigator** now includes the options that are listed in the active Teamcenter transfer option set. The transfer option set contains global search options that are applied to the entire subset search recipe. You can modify your subset search results by changing the status of one or more of these options.

Why should I use it?

You can configure the search results of a recipe in different ways without modifying the contents of your recipe.

Where do I find it?

Prerequisite	You must be in 4GD, in the subset task environment, and have an active subset search recipe.
Resource bar	**Subset Navigator**
Location in dialog box	**Recipe** group

Weld attributes in 4GD

When you create a weld design feature in 4GD, you can add custom attributes to the feature from Teamcenter using Attribute Groups and Managed Attribute Groups.

To do this, you must set the following **Welding Characteristics** customer default:

HasAttributeGroup = TRUE

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

If you change weld attributes in Teamcenter, you must load the part fully to synchronize the changes to the weld design feature.

Chapter 11: Mechatronics Concept Designer

System Navigator enhancements

The **System Navigator** is enhanced so that you can:

- Manually edit letter codes and index for logical models.
- Restore the index of logical nodes with the **Reapply Naming Rule** command.

Where do I find it?

Application	Mechatronics Concept Designer
Resource bar	**System Navigator**

Physics Navigator enhancements

The **Physics Navigator** is enhanced so that you can:

- Change the owner of physics objects between the top assembly and parts within the assembly.
- Find the owner component for an object that is selected in the physics navigator.
- Select an object in the physics navigator and make the owner component the work part.

In addition, collision bodies are now displayed as a child of the rigid body.

Where do I find it?

Application	Mechatronics Concept Designer
Resource bar	**Physics Navigator**

Exchange rigid bodies during simulations

Use the **Object Transformer** command to exchange rigid bodies during a simulation. Use this with an **Object Source** to change rigid bodies in an assembly line. Trigger the exchange using a **Collision Sensor**.

Where do I find it?

Application	Mechatronics Concept Designer

Command Finder	**Object Transformer**

Collision Body enhancements

Use **Collision Body** enhancements to do the following:

- Set collision bodies to stick together during simulations.
- Create a cylinder collision shape to detect contact between geometry during simulations.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Collision Body**

Collision Sensor enhancements

Use the **Highlight on Collision** check box to indicate when a collision sensor detects contact with a rigid body having a matching collision category.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Collision Sensor**

Joint motion restrictions

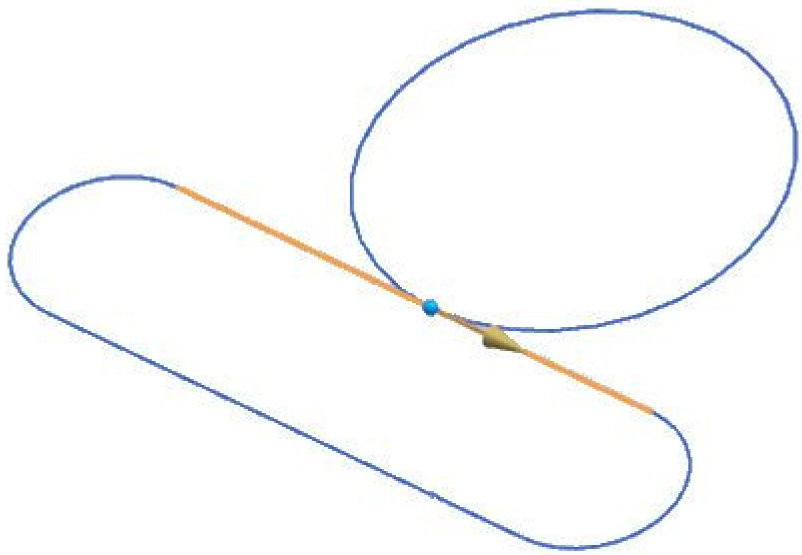
Apply motion limits directly to joints. You can specify linear or angular limits. Use this enhancement to stop angular motion after less than or greater than one full rotation.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Hinge Joint** **Sliding Joint**

Curve on Curve Joint

Use the **Curve on Curve Joint** command to create a joint for a rigid body that moves along guide curves that have a single intersection. Use gravity to initiate motion. Use the curve on curve joint to simulate objects rolling or sliding along a curve.

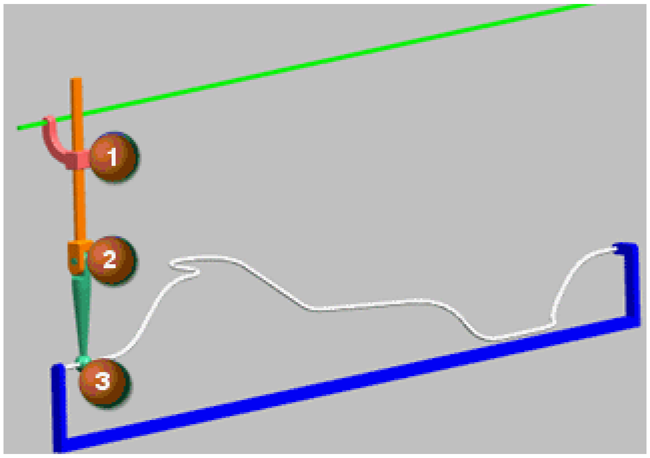


Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Curve on Curve Joint**

Point on Curve Joint

Use the **Point on Curve Joint** command to create a joint between two bodies. The attachment moves along the connected curve with the selected point being a reference that moves along the curve. The axis begins at the point and the offset is the distance between the point and zero point.



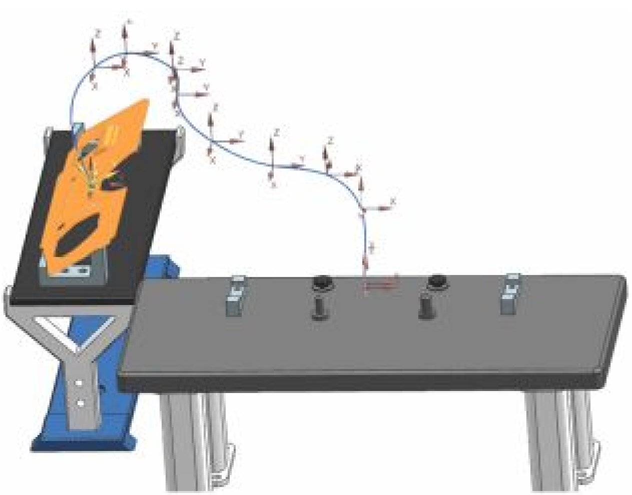
In this model, the orange bracket moves in a linear motion along the green line. A **Point on Curve Joint** is used for joint 3 so that the dark green point traces the white curve as the orange bracket moves along the green line. Joint 1 lets the point move vertically when necessary. Joint 2 lets the point move horizontally when necessary.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Point on Curve Joint**

Path constraint joint

Use the **Path Constraint Joint** command to restrict spatial motion of rigid bodies based on the desired orientation and position. You can use this to simulate the movement of parts and robotic motions.



Where do I find it?

Application	Mechatronics Concept Designer

Command Finder	**Path Constraint Joint**

Spring Damper

Use the **Spring Damper** command to apply a spring force to exert force or torque on a joint. Set the relaxed position parameter to specify the position where the force is zero.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Spring Damper**

Display Changer

Use the **Display Changer** command to change the display properties of a rigid body during a simulation. Attach a display changer to a collision sensor to set the display property of the triggered body, including color, translucency, and visibility.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Display Changer**

View simulation enhancements

Simulations are enhanced so that you can:

- Set the time scale to speed up or slow down runtime simulations.
- Create snap shots of a simulation to be used as restart points. Use this so that you do not have to view the simulation from the 00:00 time stamp.
- Increment simulations by operation, time based intervals or run simulations until a designated operation.
- View the start time, the stop time and the current time of a simulation in the sequence editor.

Where do I find it?

Application	Mechatronics Concept Designer

Command Finder	**Time Scale** **Snap Shot** **Forward One Step** **Forward One Operation**

Runtime Inspector enhancements

Use **Runtime Inspector** enhancements to do the following:

- Choose parameters that you want to graph during simulations.
- Export parameter graphical data as **.csv** files.
- Add objects to the runtime inspector view by right-clicking, and then from the list choosing **Add to Inspector**.
- View and select the simulation restart times that you created using the **Snap Shot** command. Use this so that you do not have to view simulations from the 00:00 time stamp.

Where do I find it?

Application	Mechatronics Concept Designer
Resource bar	**Runtime Inspector**

Cam Profile enhancements

Use the **Cam Profile** command to create motion relationships. The command is enhanced so that you can:

- Select different transversal segments, including third order polynomials, fifth order polynomials, sixth order polynomials, and modified acceleration sine, to connect motion segments in a cam profile.
- Use the **Optimize** command to select a transversal segment based on velocity, acceleration, or jerk characteristics.
- Edit transversal segment properties in the table view, including the transversal segment type, slopes, curvature, jerk, lambda, and optimization type.
- Use the multiple cam profile table to add, edit, and delete cam profiles in a model.

Cyclic and relative cyclic behavior

- Cyclic behavior runs the same motion profile multiple times. Use cyclic behavior when the slave value is the same at the master axis minimum and maximum value. Motion segments are not restricted by the master axis minimum and maximum value. You can wrap motion segments so that part of the segment is applied at the beginning and part is applied at the end.

- Relative cyclic behavior runs the same motion profile multiple times. The slave value at the end of the first cycle is the slave value at the start of the next cycle. Motion segments are not restricted by the master axis minimum and maximum value. You can wrap motion segments so that part of the segment is applied at the beginning and part is applied at the end.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Cam Profile**

Transport Surface enhancements

Transport Surface is enhanced so that you can:

- Apply actuators to transport surfaces as you do for joints. Applying actuators lets you control transport surfaces with operations and runtime expressions. Use the start position subgroup with position actuators to set the initial position of a conveyor.

- Use transport surfaces with curved planar surfaces.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Speed Control** **Position Control**

Electronic Cam enhancements

Electronic Cam is enhanced so that you can:

- Use a runtime parameter to control the slave axis by setting the master to **Signal**.

- Minimize disturbances on the physical axis by setting the master axis to **Virtual Axis**.

- Select an existing **Transport Surface** as the slave axis.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Electronic Cam**

Operation enhancements

Use the **Operation** command to create time and event based changes. The command is enhanced so that you can:

Export time based data

Use this to export timing and operations data including

- Step number
- Number of preceding operations
- Operation start time
- Duration of the operation
- Total time elapsed at the end of an operation
- Conditional objects and parameter names assigned to operations

Use new operator syntax

Use this to change runtime parameter values when a condition is met. Use the new operators to do the following:

- **+=** X : Use this to increment by the value X.
- **—=** X : Use this to decrement by the value X.
- ***=** X : Use this to multiply by X.
- **!** : Use this to apply a boolean NOT.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Operation**

ECAD integration enhancements

ECAD integration is enhanced so that you can:

- Associate an ECAD project to a root logical.
- View an ECAD schematic.
- Export Device Lists with Page Macro information in native NX and Teamcenter Integration.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Import from ECAD** **Export to ECAD**

SCOUT integration enhancements

SCOUT integration is enhanced so that you can import and export to **SCOUT 4.4** and **SCOUT 4.5**.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Export Cam Profile** **Import Cam Profile**

SIZER integration enhancements

SIZER integration is enhanced so that you can:

- Export data from linear or rotary actuators.
- Trigger the export of motor data with an operation or based on time.
- Import and export *.csv* or *.mdix* formats.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Export Motor Profile** **Import Selected Motors**

Shared memory signal mapping enhancements

Use the **SHM mapping** command to read and write to local memory locations. Do this to communicate with the SIMIT program. The command is enhanced so that you can:

- Select the SHM name from a list of names that are already saved in the part file.
- Set the update time unit as milliseconds, seconds, minutes or hours.
- Use automatic signal mapping by creating signals with the same data types, name and matching IO types.
- Export the signal name, type, address and initial value as a SIMIT text files.
- Set Mechatronics Concept Designer as the master during co-simulations with SIMIT.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**SHM Signal Mapping**

Tag Form and Tag Table

Tag Form

Use the **Tag Form** command to define properties for object source instances and rigid bodies. You can change the property value or assign a different physics parameter during simulation.

Tag Table

Use the **Tag Table** command to create multiple instances of a tag form. Use the tag table to set different values for each tag form instance. Use this to alternate tag form values or create parameter sequences.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Tag Form** **Tag Table**

Read-Write Device

Read-Write Device

Use the **Read-Write Device** command to assign values determined by tag forms and tag tables. Trigger the device in read mode to retrieve tag values from the rigid body. Trigger the device in write mode to assign a value to a rigid body.

Where do I find it?

Application	Mechatronics Concept Designer
Command Finder	**Read-Write Device**

Chapter 12: Line Designer

Line Designer application overview

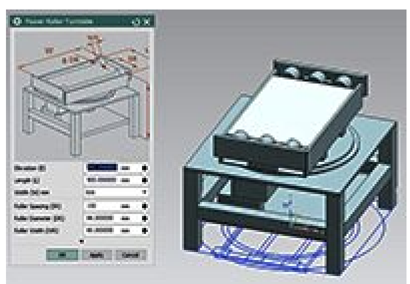
What is it?

Use the Line Designer application to quickly design and visualize layouts of production lines in NX. You can associate the production layout design to manufacturing planning using the unified Siemens platform. This close integration lets you optimize the manufacturing process by specifying each production step, down to individual manufacturing resources such as robots or fixtures.

Why should I use it?

Lay out the Production Concept

Line Designer is a complete production layout solution for manufacturing engineers. The parametric engine in NX enables you to efficiently work manufacturing components and to easily accommodate any changes. The fully classified equipment library is managed by Teamcenter, a complete solution for data and process management. Using the unified platform, you can search, view and retrieve components across a fully classified library, directly from NX.

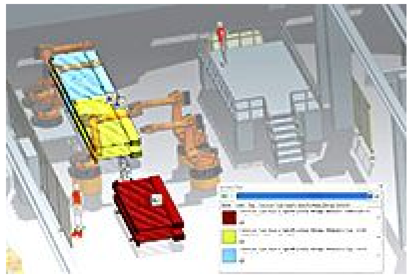


An Integrated Solution

A unified platform for product, tool, and production system design supports your entire workflow - from product and line design, and all the way to virtual commissioning. After creating a production line design with Line Designer, you can validate the manufacturing process using Tecnomatix Process Simulate. With Tecnomatix virtual commissioning solutions, you can correct PLC codes in a virtual environment before using them on real equipment.

Visual Reporting and Documentation

With HD3D Visual Reporting, you can browse PLM data in an interactive navigator to view details. Visual reports can be configured to display color-coded information about the manufacturing equipment, including component type, design changes, suppliers, investment cost and build dates. Visual reports can also be managed and distributed, to benefit the entire enterprise.



Where do I find it?

Application	Line Designer
Prerequisite	Line Designer license
Command Finder	**Line Designer**

Line Designer enhancements in NX 10.0.1

- Added support for Drag & Drop of resources from History Navigator and Teamcenter Navigator to the graphics window.
- Improved mouse button menus.
  - Removed commands that are not applicable in Line Designer 4GD.
  - Removed low-use commands; rearranged order for frequent access; added high-use commands.
  - Streamlined Smart Component shortcut menus.
- Added new command for disconnecting connected components.
- Enhanced Show/Hide and reference sets so that their state is saved with the workset.
- For 4GD mode, added the **Replace Source Part** command to the Plant Navigator context menu for resource instances. In BVR mode, the Replace Component command is applicable.
- Added an option in the connector definition for a robot that reverses the direction of propagation. If the source connector has this option enabled, then dropping the robot onto the connector of an existing instance of a riser causes the diameter from the robot to be propagated to the riser.
- Added **Type** and **Effectivity** columns to Plant Navigator. Note: In BVR mode, values in Plant Navigator columns only appear if the column is enabled in the Assembly Navigator.
- Provided a script to install Line Designer template parts to Teamcenter.

Fast placement coordinates pop-up (10.0.1)

What is it?

The **Fast Placement** capability in Line Designer has been enhanced:

- Added Z-coordinate to the coordinate pop-up.

- Added capability to edit X, Y, Z coordinate values in the coordinate pop-up.

Why should I use it?

You can use the coordinate display and editing capability to more precisely place components even while in the **Fast Placement** mode.

Where do I find it?

Application	Line Designer
Prerequisite	**Fast Placement** mode must be enabled.

Disconnect Components (10.0.1)

What is it?

The **Disconnect Components** command removes a connection between components that was created with the **Connect Components** command.

- If assembly constraints between the components exist, the assembly constraints are removed.

- Position and expression values of the components are not changed.

Why should I use it?

Disconnect components if you want to replace one of the components or change its location relative to the other component.

Where do I find it?

Application	Line Designer
Command Finder	**Disconnect Components** 

Drag and drop resources (10.0.1)

What is it?

In addition to adding resources to a layout from the Reuse Library, you can now drag and drop resources from the History Navigator and Teamcenter Navigator to the graphics window to create a new instance.

Why should I use it?

With this capability, you can take advantage of ready accessibility to existing resource instances when adding similar instances.

Where do I find it?

Application	Line Designer
Navigator	Resource instances listed in the **History Navigator** and **History Navigator**.

Chapter 13: Programming Tools

Python

What is it?

The Python programming language is added as a language binding for NX Open. This allows NX users to record and replay NX workflows in the Python language. Advanced users can create or extend NX Open Python journals to develop NX Open applications in Python. These applications can use any of the features of the Python programming language including classes, looping constructs, control statements, and callbacks. Python journals can be recorded and replayed on Windows, Linux, and Mac.

Some of the features provided with NX Open Python include:

- Block Styler support with Python code generation of saved dialog boxes
- Same NX Open API as in the other supported languages
- Wrappers implemented in Python for the NX Open UF API

Why should I use it?

You can use the Python programming language for multi-platform journal recording and replaying in NX.

Where do I find it?

Application	NX Open
Command Finder	Journal

SNAP

What is it?

Simple NX Application Programming (SNAP) is improved to provide additional capabilities when writing SNAP programs. The major enhancements are:

- Topology functions – These new functions let you get detailed information about the topology of a body (the way in which faces and edges are connected together). The topology functions help you write programs that analyze the shape and structure of solid bodies. For example, you can write programs to find holes, or fillets, or other types of features.

  Also, there are new Shell, Loop, Fin, and Vertex SNAP objects, which extend the existing Body, Face, and Edge topology objects. There is a suite of approximately 20 new enquiry functions that let you find out how these various objects are connected together. For example, you can find all the faces in a shell, all the fins meeting at a vertex, and so on.

- Temporary display functions – These functions let you draw temporary objects in the NX graphics window. You can draw markers, lines, circles, polylines, and text.

  The temporary display functions let you provide graphical prompting or feedback to the users of your programs, which often makes them easier to understand. The user can easily erase the temporary objects after they have served their purpose.

- New documentation – The *SNAP and NX Open for GRIP Users* guide is added that describes SNAP (and to a lesser extent, NX Open) from the point of view of a GRIP user. If you already know GRIP, this guide enables you to easily find corresponding functions in SNAP and NX Open.

In addition, there are other enhancements including new functions to:

- Calculate points on a B-surface very quickly
- Create curves that replicate the shapes of edges
- Analyze the convexity across an edge
- Check whether a point lies inside a body or a face
- Convert between degrees and radians
- Access the NX Work View and other standard NX views

Why should I use it?

You can write programs that can provide additional functionality and features.

Where do I find it?

The *SNAP and NX Open for GRIP Users* guide is in the *Programming Tools* section of the NX Help.

Detailed descriptions of the new functions with examples are in the SNAP Reference Guide. The new functions can be found in the following namespaces and classes:

- Topology objects and functions: new Snap.Topology namespace; additions to Snap.NX.Body, Snap.NX.Face, and Snap.NX.Edge
- Temporary display functions: Snap.Draw class
- Fast calculation of points on a B-surface: Snap.Geom.Surface.Bsurface.Position
- Create curves from edges: Snap.NX.Edge.ToCurve
- Analyze edge convexity: Snap.NX.Edge.Convexity
- Point containment: Snap.NX.Body.ContainsPoint and Snap.NX.Face.ContainsPoint
- Convert between degrees and radians: Snap.Math class
- Work view: Snap.Globals.WorkView
- NX standard views: Snap.NX.View.Standard class

UTF-8 character encoding

What is it?

NX now internally stores all character data in the UTF-8 character encoding. On Windows, the NX user interface also supports UTF-8, which enables you to set your user interface to the language of your choice. On Linux, the user interface is restricted to support characters that are in your locale and also supported by the fonts in the Ugnx10 resource file. For additional information, see UTF-8 encoding supported.

NX Open also supports the use of UTF-8 character encoding and NX Open programs can send and receive UTF-8 data. By default, NX Open supports handling data in the user's locale. To convert your program to send and receive all data as UTF-8, use the following function:

UF_TEXT_set_mode(UF_TEXT_ALL_UTF8)

When you use this function, ensure you have modified your program to properly handle UTF-8 data that is passed.

In addition, buffer lengths for most character buffers are modified to support the new NX buffer lengths for UTF-8.

Why should I use it?

You can create programs using UTF-8 character encoding.

Where do I find it?

Application	NX Open

GRIP supports internationalized characters

What is it?

GRIP now supports non-ASCII strings in addition to ASCII strings, which enables GRIP to support internationalized characters. Mixed ASCII and non-ASCII strings are also supported.

Internationalized characters are applied in NX Open GRIP programs by using a source file with a UTF-8 starting byte order mark (BOM), thereby enabling GRIP programs to use any character. The BOM is a series of bytes at the beginning of a text file that specifies the encoding of the data contained in the text file and a specific BOM signature marks the text file as containing UTF-8 data. If this signature is not present, the file contains locale data.

Without the UTF-8 BOM file, GRIP programs can still use any characters in the user's locale. However, GRIP was not enhanced to the new character-based string lengths provided for UTF-8 functionality and GRIP programs still enforce byte-based limits on data. For example, string variables are limited to 256 bytes.

The following functions are enhanced to enable GRIP programs to support internationalized characters while still providing existing behavior when dealing with ASCII data.

CHRSTR – Supports any Unicode value for a string. ASCII values still return the ASCII character.

LENF – Returns the length of a string in characters. There is no function to return the length of a string in bytes.

SUBSTR – Expects the position to be based on the number of characters and the length to extract to be based on a number of characters.

REPSTR – Uses a character count to find and search character strings, no matter how many bytes are in a character.

Why should I use it?

You can create GRIP programs using internationalized characters.

Where do I find it?

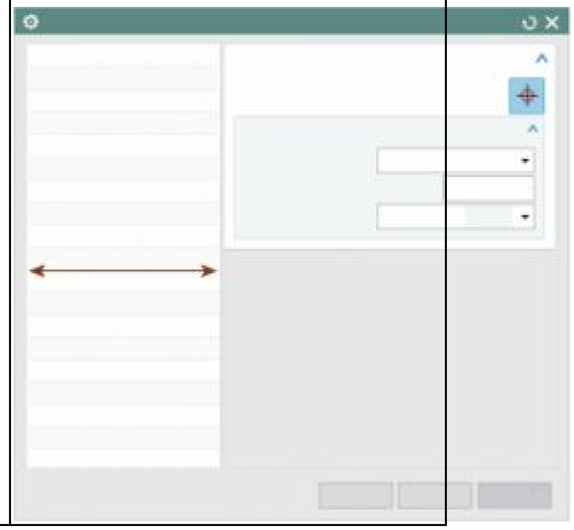
Application	NX Open GRIP

Block UI Styler

New properties for blocks

You can set additional properties for the following blocks:

Block	Property name	Description
Enumeration block	**EnableRecentOptions**	Displays recently used options at the top of the drop-down list.
Angular Dimension block	**MinRadius**	Specifies the minimum radius of the angular dimension handle that NX allows on screen. NX sets the default value of the `m_minRadius` attribute of the angular dimension interpreter object to **1.05**. The **MinRadius** property lets you override the default value.

Explorer block	**TreeWidth**	Specifies the width of the navigation tree. 

Changing a displayed part

You can use the Block UI Styler to create a dialog box which lets you change the displayed part. After you perform other operations in the new displayed part, you can return to the original displayed part.

You can choose one of many NX Open APIs to change the displayed part. Some of these APIs may delete undo marks that are set by dialog framework.

Example

The `PartCollection::SetDisplay` API deletes the undo marks set by dialog framework but the `UF_PART_set_display_part` API does not delete the undo marks.

NX supports changing the displayed part through the following APIs:

- An API which does not delete undo marks in the Update, OK, and Apply callbacks.
- An API which deletes undo marks in the OK and Apply callbacks.

If you use an API which deletes undo marks in an Update callback, you must revert to the original displayed part within the same callback. You can change the displayed part in the Update callback, and perform operations in the new displayed part within the same callback. If the displayed part is different after an Update callback execution, NX reverts to the original displayed part and closes the dialog box. NX displays the following message before closing the dialog box.

```
The displayed part was changed and the dialog’s undo mark was erased. When the undo mark is erased, a
```

If an API deletes the undo mark, NX cannot roll back to the undo mark set when the dialog box was invoked. NX cancels the dialog box and displays the following message:

```
When a dialog is cancelled, NX normally rolls back to the undo mark that was set when the dialog was
```

If you change the displayed part or perform any similar operation which deletes undo marks created by the dialog framework, what NX does depends on whether you use the **Cancel** or **OK** button to close the dialog box:

- If you use the **Cancel** button, NX cannot perform undo as the necessary undo mark is not available in NX.
- If you use the **OK** button, NX deletes all undo marks and you cannot undo operations with the **Undo** button.

Operation performed or conditions	Result when a secondary dialog box is displayed after a primary dialog box operation	Result when only the primary dialog box is displayed
	A secondary dialog box is displayed after the displayed part is changed through a callback in the primary dialog box. NX deletes all undo marks when the original displayed part is changed.	The primary dialog box remains open when displayed part is changed. NX deletes all undo marks from when the original displayed part is changed.
Cancel clicked immediately after changing the displayed part	When you click **Cancel** in the secondary dialog box immediately after changing the displayed part, NX does the following: • Closes the secondary dialog box. • Retains your original displayed part. No undo is performed for actions in the original displayed part.	When you click **Cancel** in the dialog box immediately after changing the displayed part, NX does the following: • Closes the primary dialog box. • Retains both, the original and the new, displayed parts. No undo is performed for actions in both displayed parts.
Cancel clicked in the dialog box after performing operations	When you click **Cancel** in the secondary dialog box after performing operations in the new displayed part that create new objects, or session updates, NX does the following: • Closes the secondary dialog box. • Applies Undo for operations performed in the new displayed part.	When you click **Cancel** in the dialog box after performing operations in the new displayed part that create new objects, or session updates, NX does the following: • Closes the dialog box. • Does not apply Undo for operations performed in the original or new displayed part.
OK clicked in the dialog box	When you click **OK** in the secondary dialog box, NX does the following: • Closes the secondary dialog box. • Displays the primary dialog box.	When you click **OK** in the dialog box, NX does the following: • Closes the dialog box.
Apply clicked in the dialog box	When you click **Apply** in the secondary dialog box, NX commits the actions performed and relaunches the dialog box.	When you click **Apply**, NX commits the actions performed and relaunches the dialog box.

Selection blocks in dialog box	If your primary dialog box contains selection blocks, when you open the dialog box with the original displayed part, the selection blocks display the same selection count as before.	If your dialog box contains selection blocks, the selection blocks display the selection count from the original displayed part, even if the selected objects are not available in the new displayed part. NX highlights objects selected through the selection blocks when the original displayed part is restored.

Note

The dialog framework does not support the usage of APIs that change the displayed part and delete undo marks in the OK and Apply callbacks.

Chapter 14: Browse 10.0.1 enhancements

Browse all 10.0.1 enhancements

Modeling

> Edge Blend Enhancements (10.0.1)

Manufacturing General

> IPW rendering improvements (10.0.1)
> IPW tolerances moved into the part (10.0.1)
> Generating the IPW with the operation (10.0.1)

Manufacturing Milling

> Cavity Mill Cutting Parameters (10.0.1)
> Flowcut Region Management (10.0.1)
> Corner smoothing for Z-level non-steep cut patterns (10.0.1)
> Adding center passes in Flowcut operations (10.0.1)
> Using Steep Overlap and Follow Periphery in Flowcut operations (10.0.1)
> Boundary projection for Area Milling (10.0.1)
> Minimum cut length for Area Milling (10.0.1)
> Cut region overlap in Area Milling (10.0.1)
> Divide a cut region using a line (10.0.1)
> Creating gouge free traverse moves in Zlevel operations (10.0.1)
> Create smooth noncutting motions in Zlevel operations (10.0.1)
> Tilt Tool Axis enhancements (10.0.1)
> Completely machine both sides of a blade without edge rolling (10.0.1)

Manufacturing Postprocessing

> Post Configurator (10.0.1)

Manufacturing Feature Based Machining

> Feature recognition for coaxial interrupted holes (10.0.1)
> Feature recognition enhancements (10.0.1)

Advanced Simulation

> Solver version support (10.0.1)
> Abaqus axisymmetric contact analysis (10.0.1)
> ANSYS axisymmetric contact analysis (10.0.1)
> Constraint equation support in Abaqus and ANSYS thermal solutions (10.0.1)
> CGNS results file options (10.0.1)
> Inflated laminates in LS-DYNA (10.0.1)

CMM Inspection Programming

> MCOSMOS postprocessor (10.0.1)
> CMM general enhancements (10.0.1)

Weld Assistant

> Converting legacy welds (10.0.1)

OmniCAD

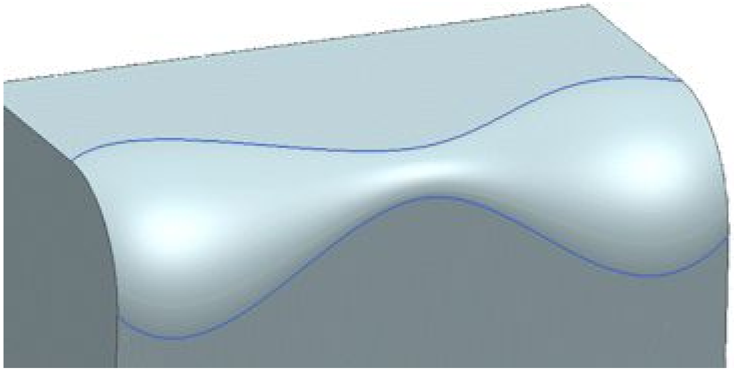
> Flow Blend (10.0.1)

Data translation

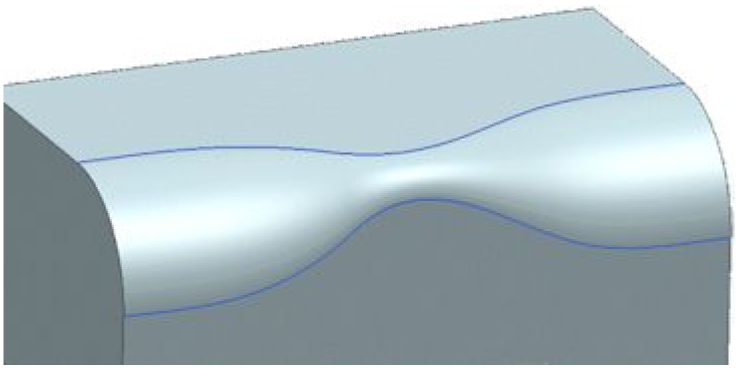
> Exporting crosshatching as grouped geometry (10.0.1)

Edge Blend enhancement (10.0.1)

When creating an **Edge Blend** with variable radius points, you can now specify that the radius function has zero slope at each end of the blended edge chain.



Zero Slope Radius Function at Chain Ends



Zero Slope Radius Function at Chain Ends

Where do I find it?

Application	Modeling
Command Finder	**Edge Blend**
Location in dialog box	**Settings→Zero Slope Radius Function at Chain Ends**

IPW rendering improvements (10.0.1)

What is it?

Needle count defaults for rendering the IPW are higher counts than in earlier releases. This is now practical due to increased performance in NX and much more capable computers than were available when the defaults were created.

Needle Count Option	Previous versions	NX 9.0.3, NX 10.0.1 and later
Coarse	200	350
Medium	350	500
Fine	500	1000

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Menu	**Menu→Preferences→Manufacturing**
Location in dialog box	**Geometry** tab→**In Process Workpiece Tolerances**

IPW tolerances moved into the part (10.0.1)

What is it?

When you have a part that requires IPW settings that are different from those set by the **IPW Tolerances** group in the customer defaults, **Chordal Tolerance**, **Tolerance Mode**, **Needle Distance**, and **Needle Count**, you can now override the defaults in the **Manufacturing Preferences** dialog box. These settings are saved with your part.

The customer defaults now serve only to establish default tolerance values for the **Manufacturing Preferences** dialog box.

Why should I use it?

Part-specific preferences are useful when you have parts that require special settings, for example a few large parts that require a higher needle count than your other parts.

You can change the IPW tolerances and see the effect without having to restart your NX session.

You can create template parts that have specific settings for the various types of parts with which you deal regularly.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Menu	**Menu→Preferences→Manufacturing**
Location in dialog box	**Geometry** tab→**In Process Workpiece Tolerances**

Generating the IPW with the operation (10.0.1)

What is it?

A new customer default, **Generate IPW with Path**, lets you generate the in-process workpiece (IPW) as you generate an operation. NX saves the IPW with the operation.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

Why should I use it?

Saving the IPW with the operation saves time when you do the following:

- Use the **Verify Tool Path**→**3D Dynamic** method.
- Select the **Show Thickness by Color** option or the **Workpiece**→**Show 3D** option.
- Generate operations that use the previous IPW.

You can:

- Visually inspect the operation result as soon as it is generated.
- Immediately verify the tool path to check for collisions with the IPW.

Where do I find it?

Initial release	NX 9.0.3 MP2
Application	Manufacturing
Customer Defaults	Search  for **Generate IPW with Path**

Cavity Mill Cutting Parameters (10.0.1)

What is it?

When you create a tool path that uses the **Follow Periphery** cut pattern, the **Legacy** options in the **More** tab in the **Cutting Parameters** dialog box are no longer available when you set the **Pattern Direction** to **Automatic**.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Location in dialog box	**Cutting Parameters** dialog box→**Strategy** tab→**Pattern Direction** list→ **Automatic**

Flowcut Region Management (10.0.1)

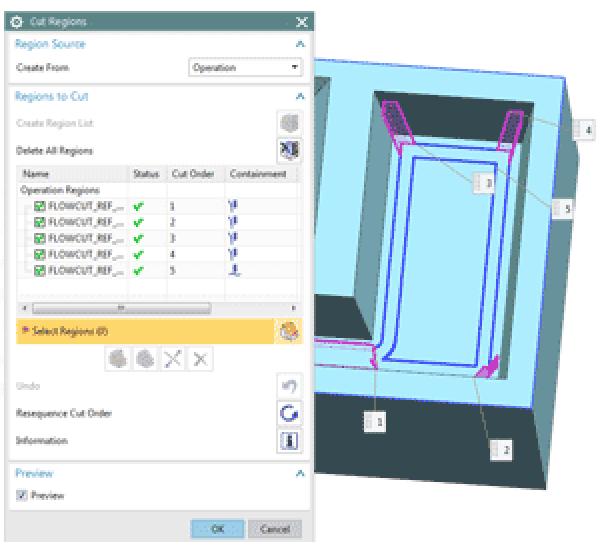
What is it?

In a Flowcut operation, you can create, visualize, edit, and save cut regions without generating the tool path. **Cut Region** options will be available in the **Flow Cut** geometry group.

You Can:

- Generate the tool path based on the saved cut regions within the operation.
- View the regions graphically.
- Import inactive regions from previous sessions into the current session.

When changes are made to the parameters of a region, NX displays a message and marks the region with an Invalid icon in the region list until the operation is regenerated.



Why should I use it?

Because you can edit cut regions before you generate the operation, overall processing time can be reduced.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Flowcut Operation
Location in dialog box	**Flow Cut Operation** dialog box

Corner smoothing for Z-level non-steep cut patterns (10.0.1)

What is it?

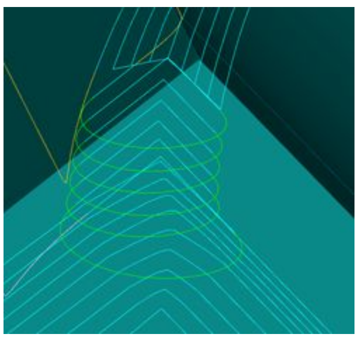
You can smooth corners in **Flow Cut** operations with **Zig**, **ZigZag**, or **ZigZag with Lifts** patterns for non-steep regions. To do this, you:

1. Set steep regions to **Zlevel**.

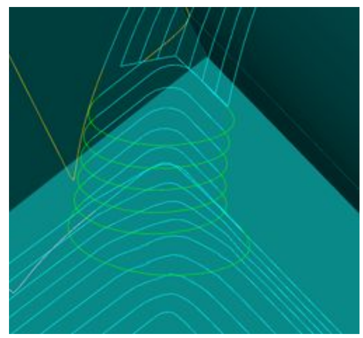
2. Set the **Smoothing** cutting parameter to **All Passes**.

3. Specify a radius size to be applied to the corners.

4. Generate the operation.



☐ **Corner Smoothing**



☑ **Corner Smoothing**

Why should I use it?

When you machine hard material, or machine at high speeds, consider adding fillets to all corners. Corner fillets can do the following:

- Prevent a sudden change in direction and cutter deflection which can cause excessive stress on the machine tool and cutter.

- Aid in tool path generation for Nurbs output because smooth transitions are easier to blend into Nurbs than sharp corners.

The **Smoothing** option gives you better control of tool path output for finishing operations and improves machining efficiency.

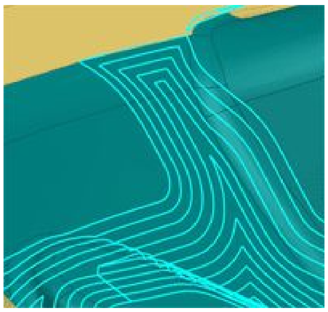
Where do I find it?

Initial release	NX 9.0.3 MP2
Application	Manufacturing
Location in dialog box	**Flowcut Reference Tool** operation→**Cutting Parameters**→**Corners tab**→**Path Shape in Corners** group→**Smoothing**→**All Passes**

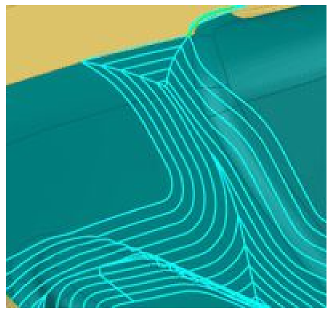
Adding center passes in Flowcut operations (10.0.1)

What is it?

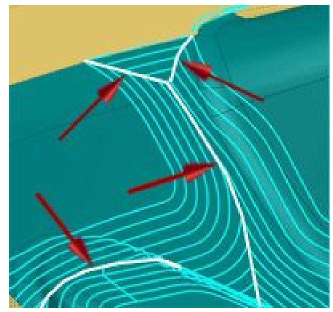
When you machine corners in Flow Cut operations, you can reduce or remove scallops. To do this, use the **Add Center Passes** option. This option creates a center pass in the region when space between offset passes produces a larger than desired gap that results in a scallop.



☐ Add Center Passes



☑ Add Center Passes



Added paths highlighted.

Why should I use it?

This option helps you to improve surface finish when you machine corners.

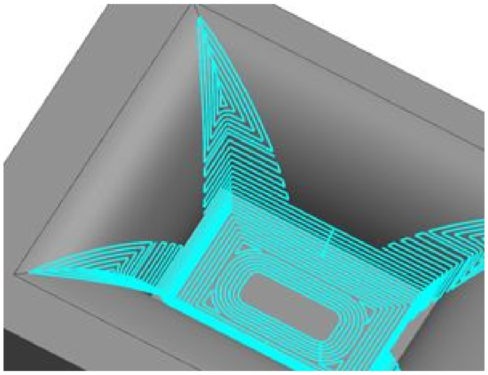
Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Flowcut Operation
Location in dialog box	**Flow Cut Drive Method** dialog box→**Non-steep Cutting** and **Steep Cutting** groups

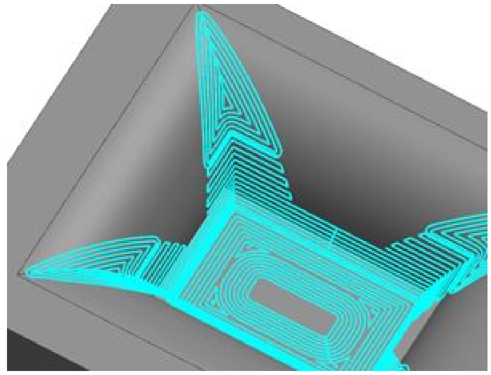
Using Steep Overlap and Follow Periphery in Flowcut operations (10.0.1)

What is it?

You can overlap the cutting motions between steep and non-steep regions in Flow Cut operations. Use the **Steep Overlap** option to specify the distance or a percentage of the tool diameter overlap. Additionally, you can use a Follow Periphery pattern for both steep and nonsteep regions.



No **Steep Overlap**



Steep Overlap set at 0.100”

Why should I use it?

When you overlap the cut motions, you can sometimes achieve a smoother finish.

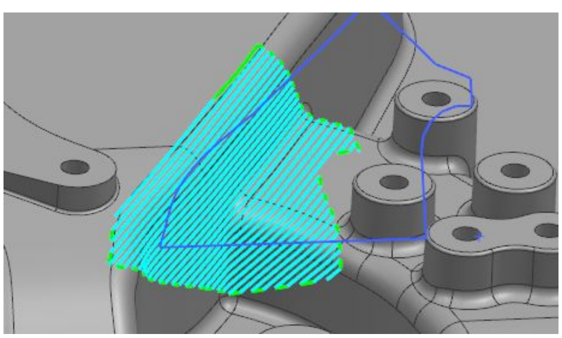
Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	**Flowcut** operation using a **Zlevel** steep pattern with a **Follow Periphery** non-steep pattern.
Location in dialog box	**Flow Cut Drive Method** dialog box→**Steep Cutting** group

Boundary projection for Area Milling (10.0.1)

As you define or edit a boundary with the **Tool Position** option set to **Contact**, the outer boundary of the contact position is dynamically calculated and shown on all surfaces through the part.

The following images show arbitrary curves used to define a boundary that contains the cutter within a small area of the part.



The boundary curves contain the toolpath in the area shown.

When the **Trim Boundaries** dialog box is open, and the **Tool Position** option is set to contact, NX displays the tool contact boundary on all part surfaces. The boundary that is displayed on the surface that is being cut shows you the actual cut area.

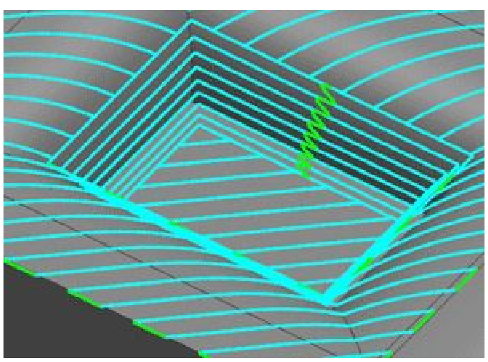
Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisites	Operations must use the **Area Milling** drive method. In geometry groups, the **Geometry Subtype** option must be set to **MILL_AREA**. In the **Trim Boundaries** dialog box, in the **Boundaries** group, the **Tool Position** option must be set to **Contact**.

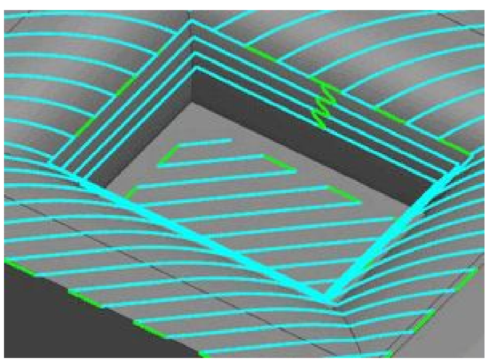
Minimum cut length for Area Milling (10.0.1)

What is it?

In Area Milling operations, you can now specify a minimum cut length for the steep Zlevel pattern.



Minimum cut length: 0.000”



Minimum cut length: 2.300”

Use the **Minimum Cut Length** option to determine which toolpaths to keep in the current operation. NX removes tool path cutting motions that are smaller than your specified minimum cut length, and cuts the Zlevel pattern with no holes or gaps.

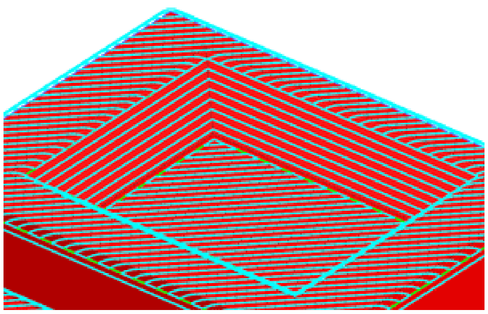
Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Area Milling operation
Location in dialog box	**Area Milling Drive Method** dialog box→**Drive Settings** group→**Steep Cutting**

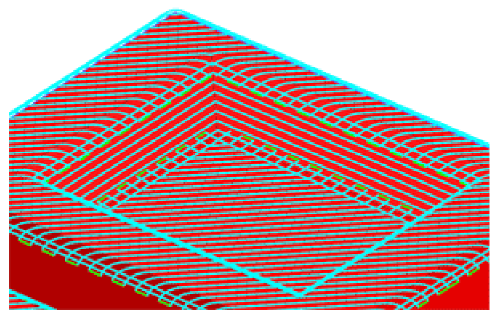
Cut region overlap in Area Milling (10.0.1)

What is it?

You can overlap the cutting motions between steep and non-steep regions in Area Milling operations. Use the **Overlap Steep and Non-Steep Regions** option to specify the distance by which the regions overlap.



Zero overlap applied



A .050” overlap applied

In the example shown, a .050” overlap has been applied. The .050” distance is divided by 2 and the regions are extended towards each other by .025” while following the part geometry.

Why should I use it?

When you overlap the cut regions, you can sometimes achieve a smoother finish.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	**Area Milling** operation
Location in dialog box	**Area Milling Drive Method** dialog box →**Steep Containment** group →**Overlap Steep and Non-steep Regions**

Divide a cut region using a line (10.0.1)

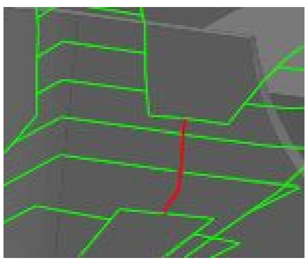
What is it?

In Area Milling operations, you can divide a cut region using a line with additional options. Use the Divide by Line option to create and control the placement of the line. You can:

- Create points that NX uses to create the line.
- Project the line on to the part.
- Preview the division. If the regions are not what you expect, you can change the line.

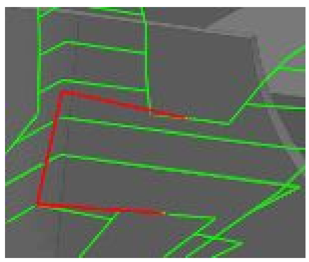
After you create the line, NX can create a plane that slices through the line and divides the cut region.

To specify how you want the plane to be oriented, use one of the following **Divide Plane** options:



Tool Axis

Use this option to project the plane is projected through the current tool axis.

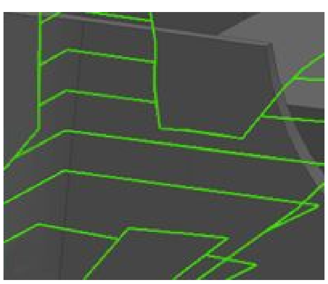


Normal to Tool Axis

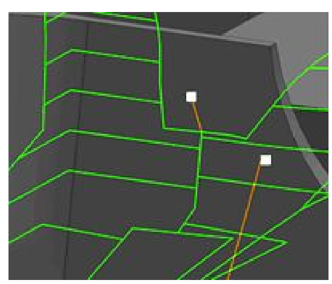
Use this option to project the plane perpendicular to the tool axis.

Dynamic

Use this option if you want to project the plane using an angle that you specify. You can either enter a value or drag the angel handle. The angle is referenced from the tool axis.



The cut region before any divisions.



The cut region after using the **Tool Axis Divide Plane** option.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing

Prerequisite	Area Milling operation
Location in dialog box	[Area Milling operation] **Cut Regions** dialog box

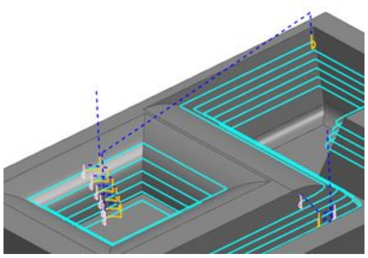
Creating gouge free traverse moves in Zlevel operations (10.0.1)

What is it?

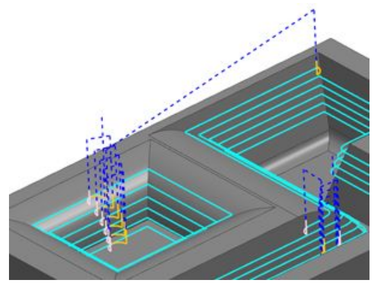
In Zlevel operations, for traverse moves, you can now avoid potential collisions between the cutter and the part or IPW. To do this, you set a safe clearance value and then apply it.

- To set the value, use the **Safe Clearance Distance** option.
- To apply the value, use the **Apply Safety Clearance to Direct** option.

If you do not set a **Safe Clearance Distance** value, NX notifies you that a potential collision could occur.



☐ Apply Safety Clearance to Direct



☑ Apply Safety Clearance to Direct

A 0.300” distance is used.

Option name change

In the **Non Cutting Moves** dialog box, on the **Transfer/Rapid** tab, on the **Transfer Type** list, the **Lowest Safe Z** option is now named **Direct / Previous Plane Backup**.

Where do I find it?

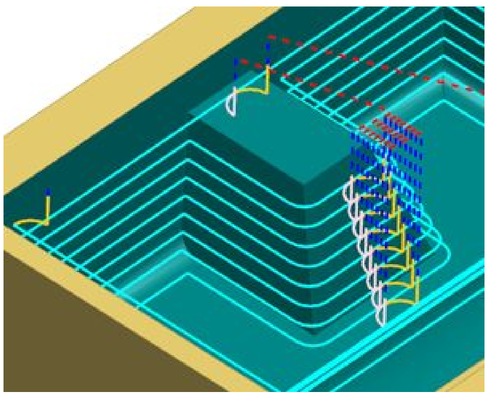
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Zlevel operation
Location in dialog box	[Zlevel operation]→**Path Settings**→**Non Cutting Moves**→**Transfer/Rapid** tab→**Between Regions** and **Within Regions** groups

Create smooth noncutting motions in Zlevel operations (10.0.1)

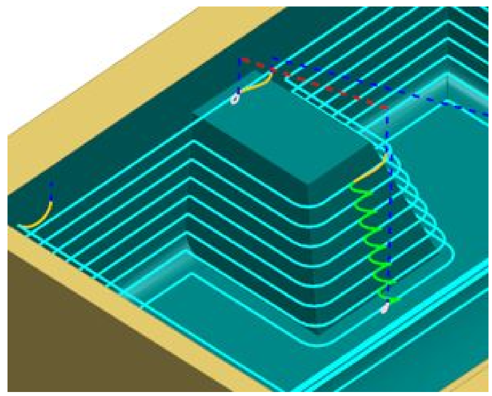
What is it?

You can override the engage, retract, and stepover motions within a region for Zlevel operations. To do this, use the **Override with Smooth Connections** option on the **Smoothing** tab. NX outputs a smooth continuous motion along a spline that is generated between cutting motions. You can specify the following parameters on the **Smoothing** tab:

- The length and height of the smooth engage and retract motions
- Maximum stepover and part safe clearance
- The distance of the smoothing region
- The tolerance



☐ Override with Smooth Connections



☑ Override with Smooth Connections

When machining electrodes for example, NX will apply safe clearances to Negative Part Stock values for traverse moves. NX also creates consistent, smooth stepover lifts off the part for each pass. If a smooth engage is not possible, NX notifies you and uses a plunge motion instead.

If you are not satisfied with the automatic start point of the tool path, you can specify the start point for the region on the **Start/Drill Points** tab.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Zlevel operation
Location in dialog box	**Non Cutting Moves** dialog box→**Smoothing** and **Start/Drill Points** tabs

Tilt Tool Axis enhancements (10.0.1)

What is it?

There are new usability enhancements that keep the latest tool axis settings stored in the following cases:

- If you generate a tool path using the **OK** command, and then stop it.
- If you generate a tool path and then use the **UNDO** command.

The **Tilt Tool Axis** dialog box now groups the options logically on the **Tilting**, **Clearance**, and **Machine Tool Characteristic** tabs.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Command Finder	**Tilt Tool Axis**

Completely machine both sides of a blade without edge rolling (10.0.1)

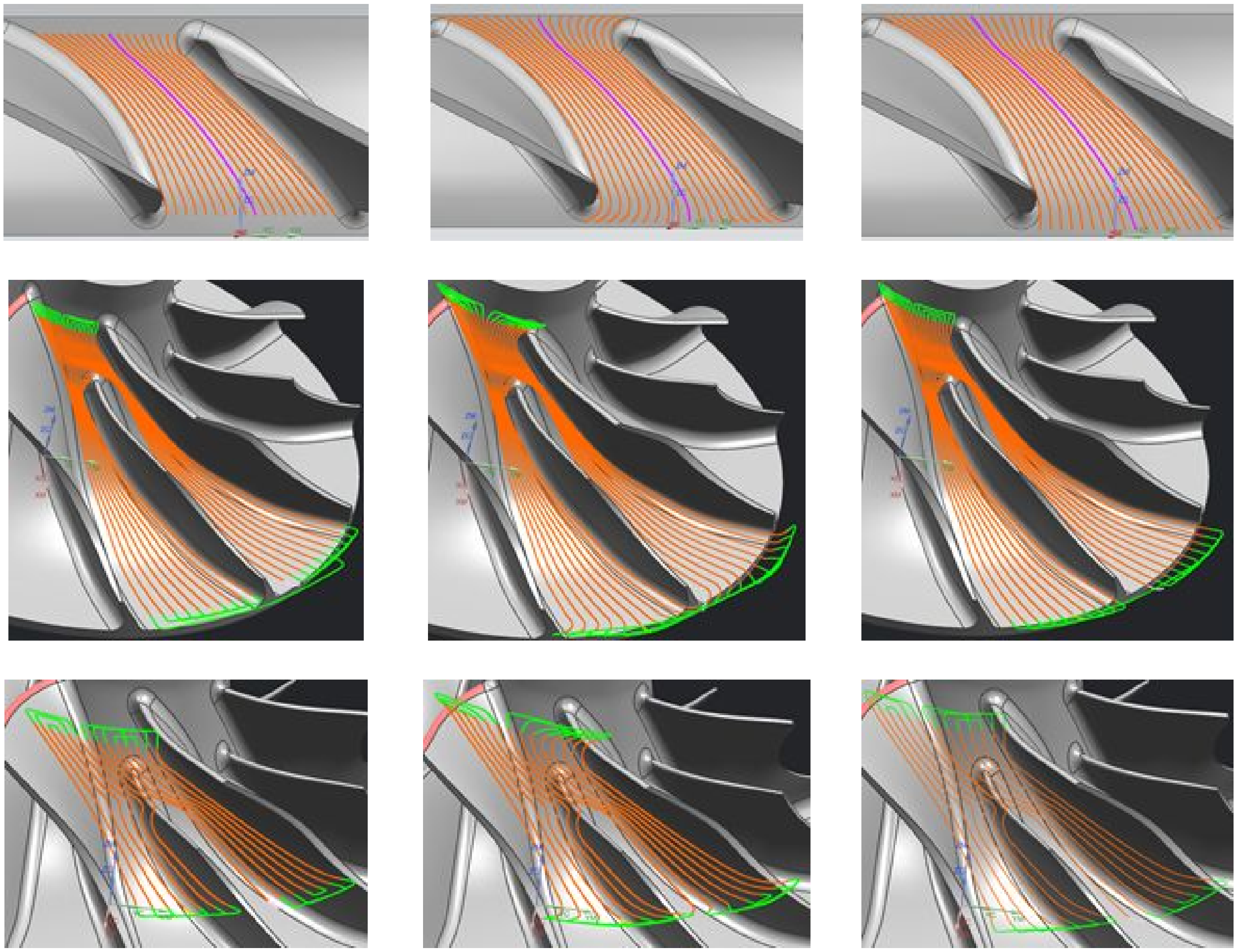
In multi blade operations, the **Blade Edge** option controls where the path stops following the blade geometry, and begins extensions that are not driven by the blade shape. You can now use the new **No Curling** edge trimming option to prevent the cutter from curling over the blade edges. This is helpful in the following cases:

- For blades with sharp corners at the trailing edge, the path follows the pressure and compression sides, and extends tangentially without any curling around the sharp corners.
- For blades that are highly curved around the part axis, NX prevents fast changes in the path direction as the path moves from the pressure side to the leading edge.
- For operations with the cut levels depth mode set to **Offsets from Hub**, when a cut level reaches the shroud height, NX extends the path beyond the top edge of the blade without curling around the blade.

For roughing and hub finish operations, NX automatically adds a small tangential extension so that all passes of a cut level extend the same distance from the rotary axis.

For all operations, you can also enter an **Extension** value to extend the path tangentially. NX adds any user specified extension after the automatic extension.

The following examples show how the **No Curling** option automatically controls the tool path at the leading and trailing blade edges. The stepovers are exaggerated to show detail.



Blade Edge = Along Blade Direction

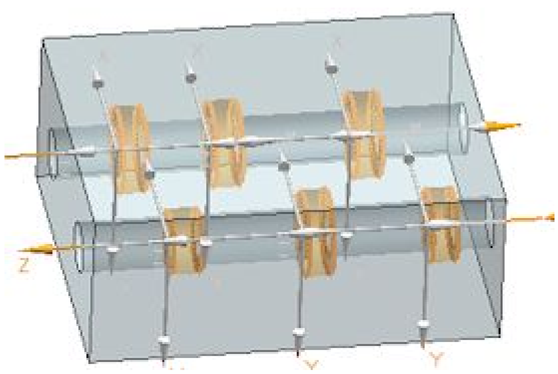
Blade Edge = Along Part Axis

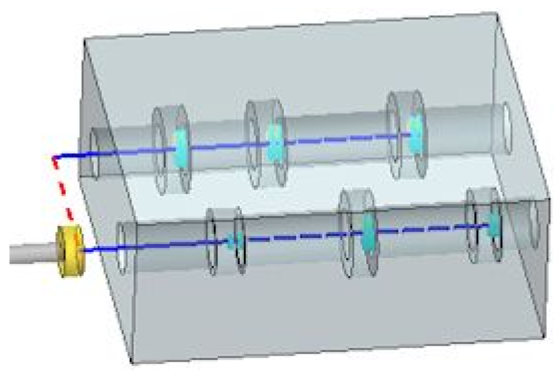
Blade Edge = No Curling

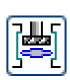
Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Multi Blade operation
Location in dialog box	[Multi blade operation] dialog box→**Drive Method** group→edit drive method  Blade rough, hub finish operations: **Multi Blade Rough Drive Method** or **Hub Finish Drive Method** dialog box→**Leading Edge** group Blade finish operations: **Blade Finish Drive Method** dialog box→**Cut Periphery** group→**Leading Edge** subgroup Blend finish operations: **Blend Finish Drive Method** dialog box→**Cut Periphery** group→**Trailing Edge** subgroup

Radial Groove Milling (10.0.1)





The **Radial Groove Milling**  operation combines groove milling and hole milling to machine a sequence of circular grooves with a T-cutter. The operation cuts in a circular pattern with multiple radial and axial passes.

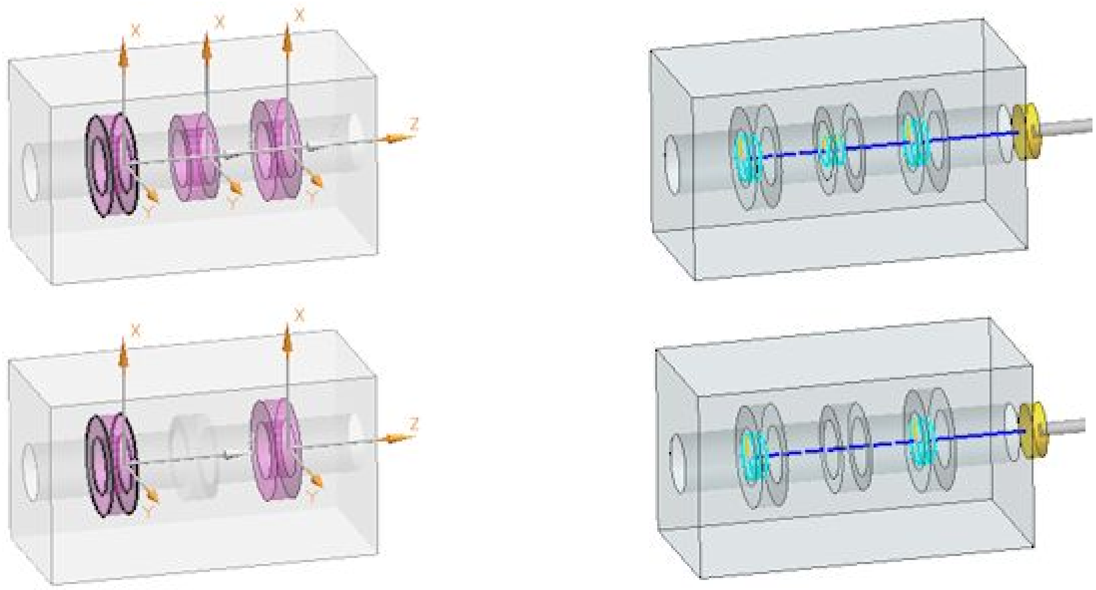
Your workflow can include multiple operations. Create a roughing operation that leaves stock on the floor, ceiling, and side, and then remove the remaining material in a subsequent finishing operation.

You can manually select the groove geometry from within the operation, or use previously recognized groove features within a feature group. NX updates the tool path when you regenerate if there are:

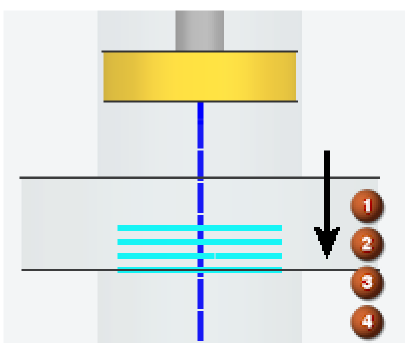
- Dimensional changes

- Sequence changes

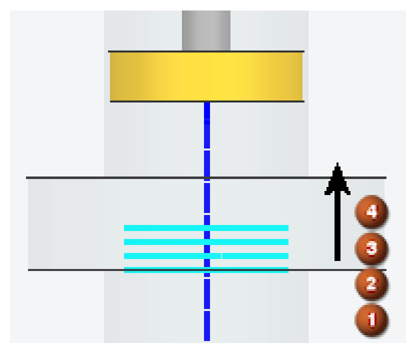
  You can add grooves to the sequence or remove grooves from the sequence.



The **Radial Groove Milling** operation uses the same level sequencing options that are available in **Groove Milling**. For **Radial Groove Milling** operations, the ceiling is the face closest to the direction that the tool approaches from.



Ceiling to Floor



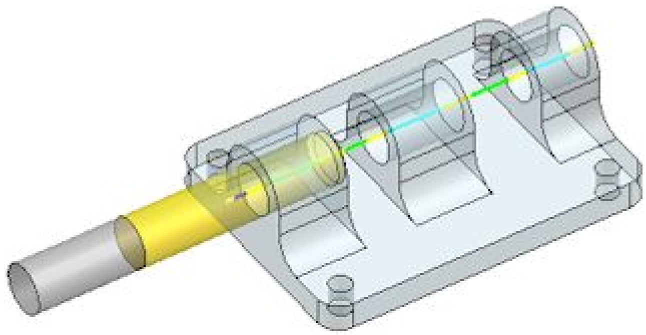
Floor to Ceiling

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	T-cutter tool, enclosed groove
Command Finder	**Create Operation**
Location in dialog box	**Create Operation** dialog box→**Type** list→**hole_making**→**Operation Subtype** group→**Radial Groove Milling**

Drilling interrupted holes (10.0.1)

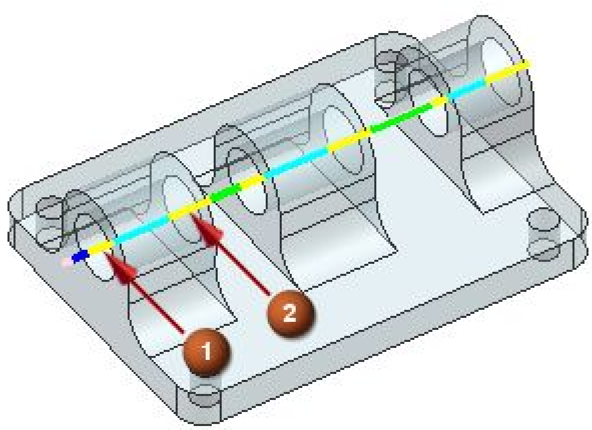
Use the **Sequential Drilling**  operation to drill a sequence of coaxial holes.



You can:

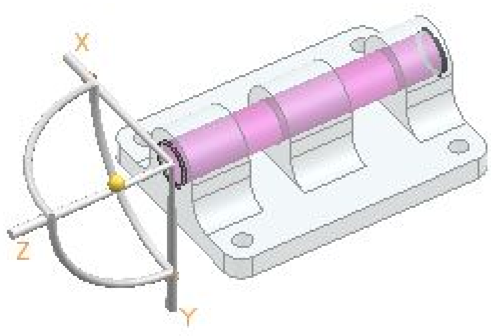
- Create a feature group geometry parent first, or specify the geometry from within the operation.
- Set the cycle tracking point.
- Drill with or without chip break.

  For the chip break cycle, you can also specify a clearance and retract for each hole segment.
- Slow the feed rate when the tool enters material, and again when the tool breaks through material. Use the **First Cut** and **Last Cut** options to specify the distance and feed rates for entering (1) and leaving (2) material.



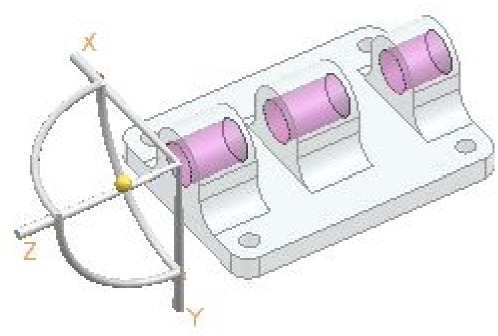
Interrupted hole features

For a **Sequential Drilling** operation, the **Motion Output** option is set to **Interrupted Moves**. This tells NX to ignore empty spaces when it creates the feature geometry.



Hole feature

Motion Output = **Machine Cycle**



Hole feature

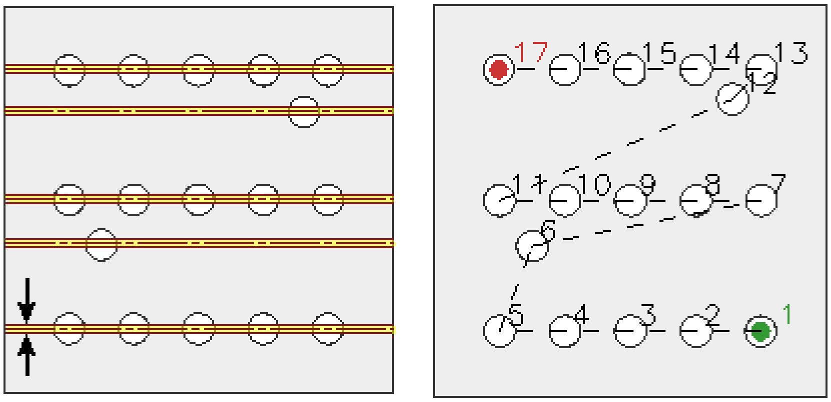
Motion Output = **Interrupted Moves**

Where do I find it?

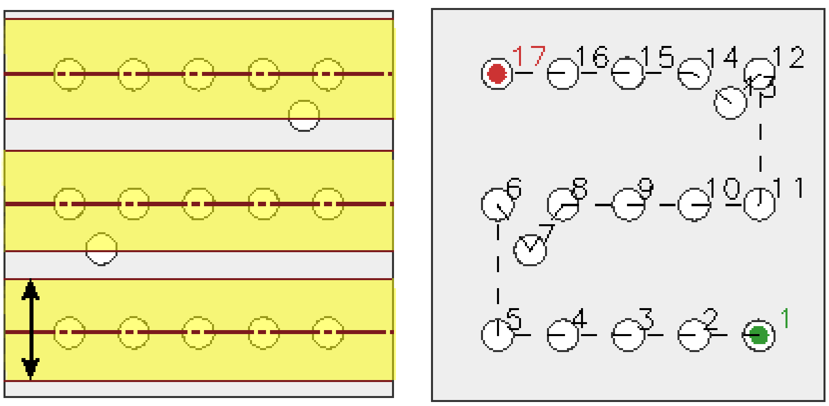
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Command Finder	**Create Operation**
Location in dialog box	**Create Operation** dialog box→**Type** list→**hole_making**→**Operation Subtype** group→**Sequential Drilling**

Controlling the width for zig and zig zag hole patterns (10.0.1)

When you optimize a machining sequence, the default tolerance that NX uses for zig and zig zag hole patterns can create additional passes when the holes are not aligned. Use the **Band Width** option to specify a width that is large enough to include nearby holes in each pass. NX applies half of the specified value to both sides of the pass center line.



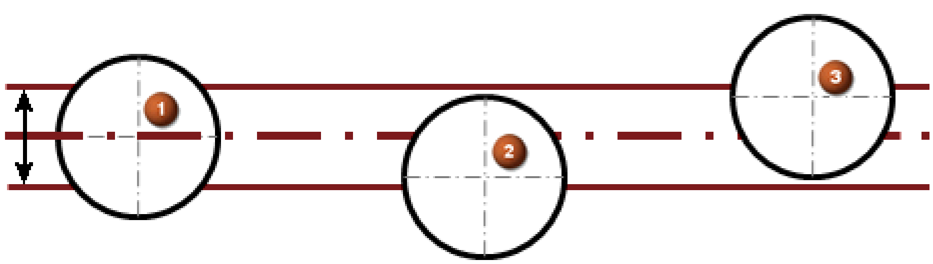
Band Width = 1 mm (default value)



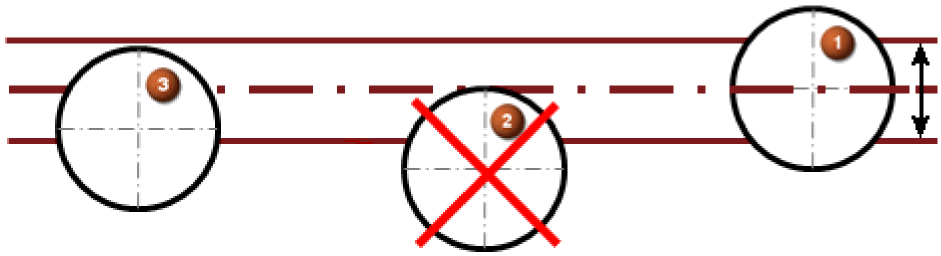
Band Width = 40 mm

The first hole in each pass determines the center line for the pass. NX calculates the band width from this center line, and the band width determines which holes to include in the pass. If you reverse the machining order, a hole that was previously included in the pass could be excluded. NX machines only those holes whose center lines fall within the band width.

In this example, the center lines of holes 2 and 3 are within the band width for the pass. NX includes holes 1, 2, and 3 in the pass.



When the order is reversed, the center line of hole 2 is not within the band width for the pass. NX includes only holes 1 and 3 in the pass.



Note

You must enter the **Band Width** value each time you optimize the sequence.

The options in the **Sequence** group in the **Hole or Boss Geometry** and **Feature Geometry** dialog boxes always display the default settings. The sequencing options are *actions* that NX executes when you click **Reorder List**. NX saves the resulting sequence without saving the settings.

Where do I find it?

Initial release	NX 9.0.3 MP2
Application	Manufacturing
Prerequisite	Hole machining sequence with the **Optimization** option set to **Primary Direction**.
Location in dialog box	**Hole or Boss Geometry** or **Feature Geometry** dialog box→**Sequence** group→**Band Width**

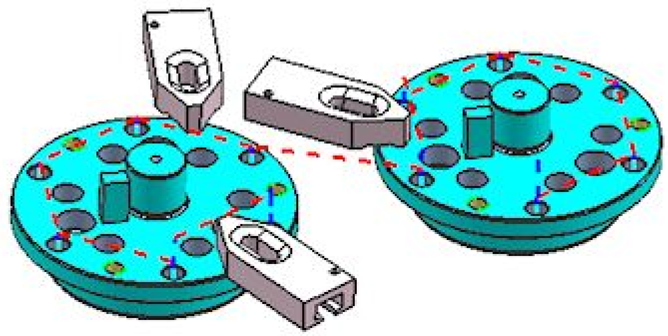
Reduce transition motions and machining time for drilling programs (10.0.1)

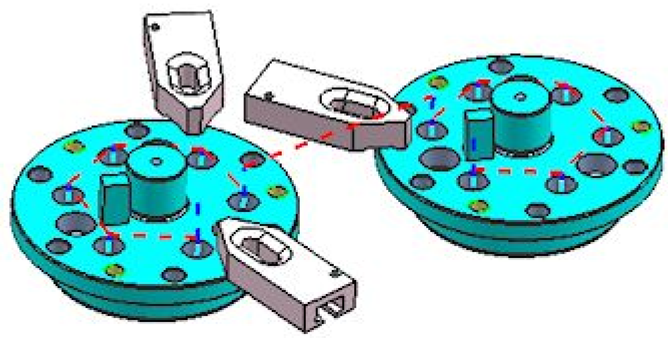
You can now optimize drilling operations across multiple features and workpieces to minimize transition motions and machining time within a program.

- Reorder operations to minimize tool changes.
- Create optimization groups for further optimization and verification.
- Apply operation level sequencing at the optimized group level.
- Generate and verify tool paths at the optimized group level. The **Optimize NC** dialog box includes the standard operation tool path actions.

Before optimizing

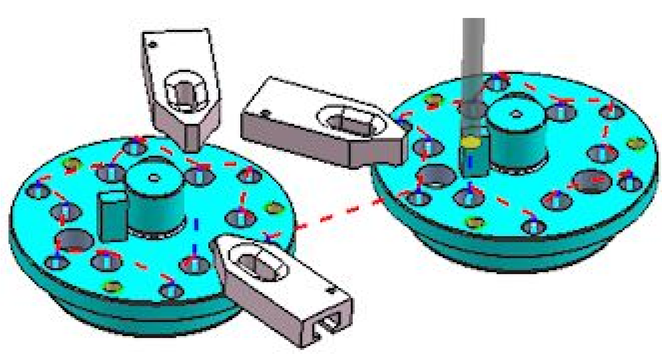
Operation Navigator, **Geometry** view	**Operation Navigator**, **Program Order** view
WORKPIECE **FG_STEP1HOLE** **SPOT_DRILLING** **DRILLING_UP** **DRILLING_FINISH** **FG_STEP1HOLE_STEP1POCKET** **SPOT_DRILLING_2** **DRILLING_UP_2**	**PROGRAM** **SPOT_DRILL** Contains two operations **SPOT_DRILLING** **SPOT_DRILLING_2** **DRILL** **DRILLING_UP** **DRILLING_FINISH** **DRILLING_UP_2**





After optimizing

Operation Navigator, **Geometry** view	**Operation Navigator**, **Program Order** view
WORKPIECE **FG_STEP1HOLE** **SPOT_DRILLING** **DRILLING_UP** **DRILLING_FINISH** **FG_STEP1HOLE_STEP1POCKET** **SPOT_DRILLING_2** **DRILLING_UP_2**	**PROGRAM** **SPOT_DRILL** **OPTIMIZED_NC** **SPOT_DRILLING** **SPOT_DRILLING_2** **DRILL** **DRILLING_UP** **DRILLING_FINISH** **DRILLING_UP_2**



Where do I find it?

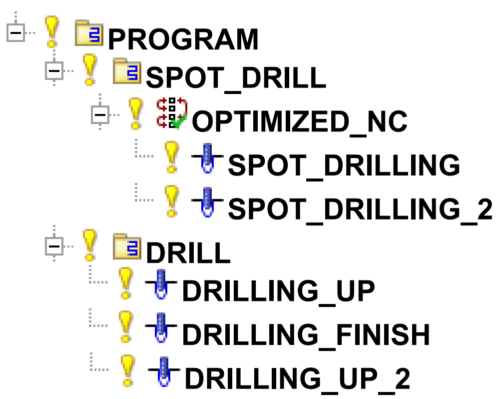
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Operation Navigator	Right-click the program→**Object**→**Optimize**

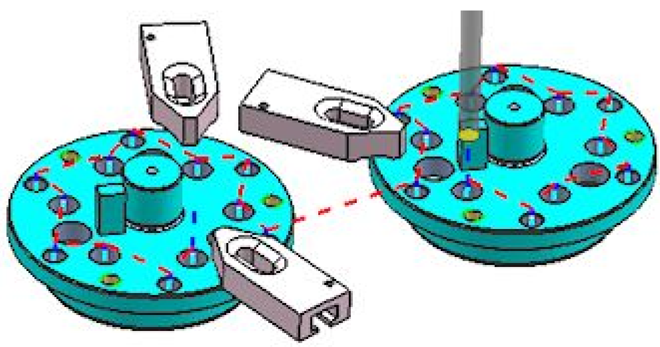
Controlling non cutting moves for an optimized program group (10.0.1)

When you optimize multiple drilling operations at the program level, NX creates an optimized program group, **OPTIMIZED_NC**. You can now specify non cutting moves for this program group, and these moves override the non cutting moves defined in the individual operations.

By default, NX uses the non cutting moves defined in the first operation for the optimized program group. You can use the default settings, or change the non cutting move settings for this program group.

Operation Navigator, **Program Order** view





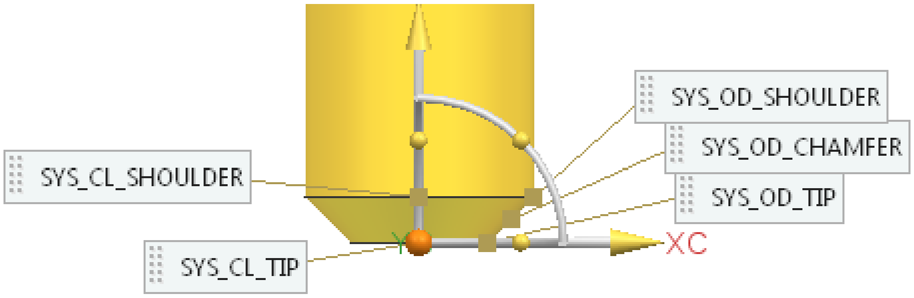
Where do I find it?

Initial release	NX 10.0.1
Application	Manufacturing
Operation Navigator	Right-click the program→**Object**→**Optimize**
Location in dialog box	**Optimize NC** dialog box→**Non Cutting Moves**

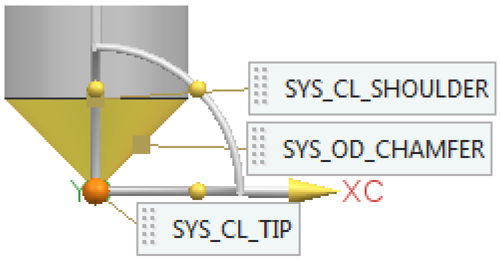
Tracking points for chamfering tools (10.0.1)

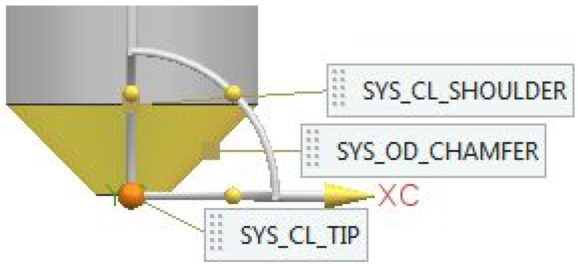
A tracking point is the internal reference point on the tool that NX uses to calculate tool paths.

Chamfer milling tools have two system-defined tracking points at the center line and three system-defined tracking points at the shoulder.



Spot drills and countersinking tools have an added tracking point in the middle of the chamfer.





You can also define additional tracking points anywhere on the tool to use in your operation.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Chamfer milling tool
Location in dialog box	Chamfer milling tool dialog box→**More** tab→**Tracking** group→**Tracking Points**→**Tracking Points** dialog box

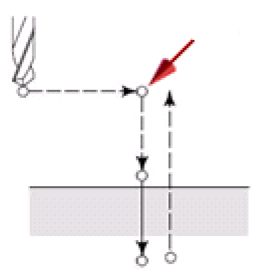
Controlling retract moves for canned drilling cycles (10.0.1)

You can choose how NX outputs the retract moves for the canned cycles in a drilling operation. To do this, use the **Retract Output Mode** option. Select the setting that matches the output to your controller requirements.

- **Clearance Only** outputs the retract level as part of the canned cycle only when the retract level is equal to the clearance level. In all other cases, NX turns off the cycle and generates an explicit move to the calculated retract point.
- **Clearance Initial** outputs the retract level as part of the canned cycle when the retract level is equal to the clearance level or equal to the initial level.
- **Always** outputs the retract level as part of the canned cycle for postprocessors that can handle this input. The cycle includes the final retract.

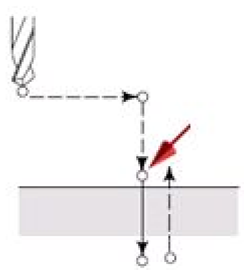
When the tool reaches the bottom of a hole, NX returns the tool to one of the following positions:

1. The initial level, which is the G98 position.



   The initial level is defined as the last Z position before NX begins the cycle.

2. The defined R-plane, which is the G99 position.



The R-plane is the plane at which the drilling operation begins. NX defines the R-plane and the clearance plane relative to the cycle level. For a constant clearance, if the cycle level changes, so does the R-plane.

Note

The initial level does not change when drilling in G99 mode.

3. A Z value on or above the R-plane that is based on the noncutting moves settings.

The following examples show how the **Retract Output Mode** option affects the tool paths when using the **Lowest Safe Z**, **Clearance – Tool Axis**, and **Direct** options for transferring between holes. The holes are machined from left to right, omitting holes 7, 8, 15, and 16.

- Yellow = Departure
- Blue = Approach
- Green = Explicit traverse outside of the cycle
- Red = Implicit traverse within the cycle

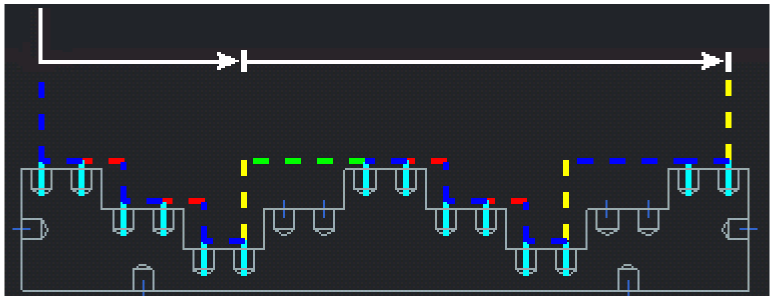
Lowest Safe Z

For the **Lowest Safe Z** examples, at the 6th hole, the retract level is below the clearance level, and also below the initial level. At the 14th hole, the retract level is below the clearance level, but at the initial level.

For the **Clearance Only** option, NX outputs a `CYCLE/OFF` statement at the 6th, 14th, and final holes.

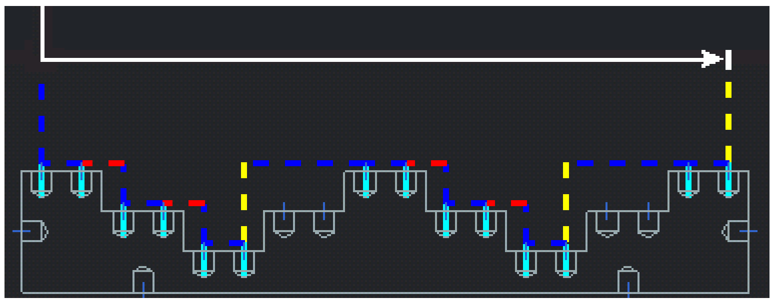
Clearance Only, 3 cycles

For the **Clearance Initial** option, NX outputs a `CYCLE/OFF` statement at the 6th and final holes.



Clearance Initial, 2 cycles

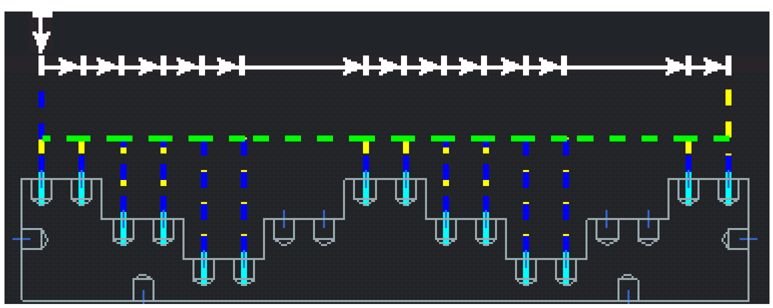
For the **Always** option, NX outputs a `CYCLE/OFF` statement at the final hole.



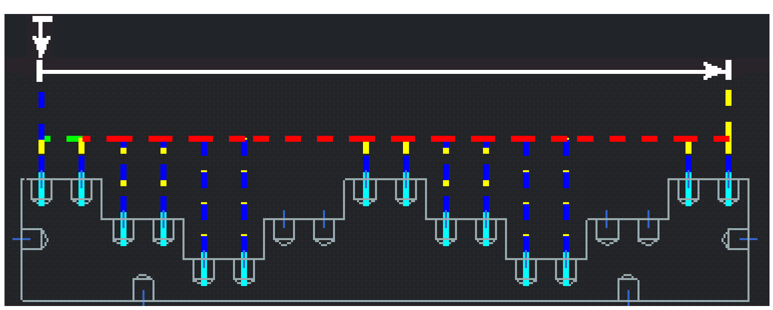
Always, 1 cycle

Clearance – Tool Axis

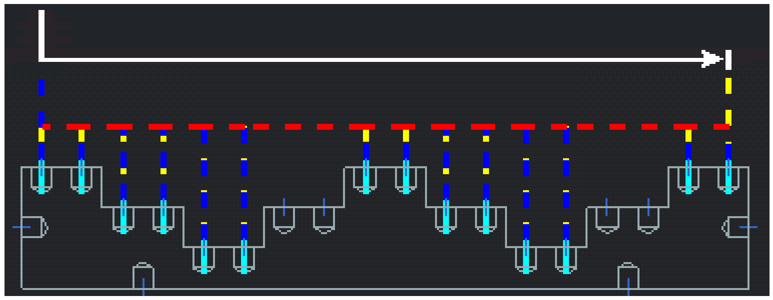
For these examples, the clearance is to the bounding box.



Clearance Only, 14 cycles

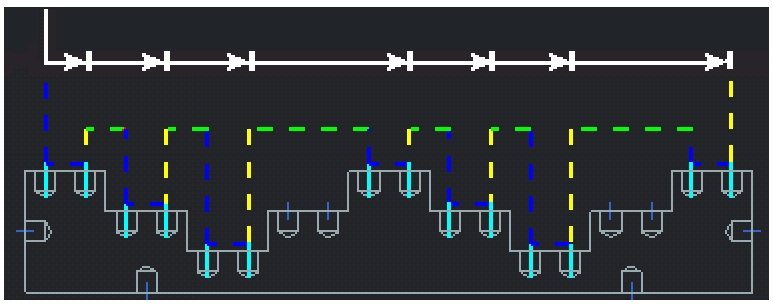


Clearance Initial, 2 cycles

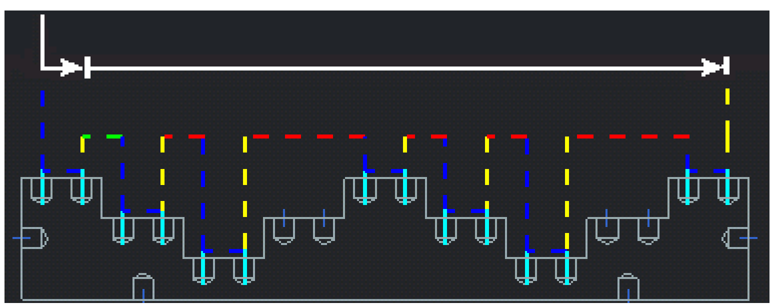


Always, 1cycle

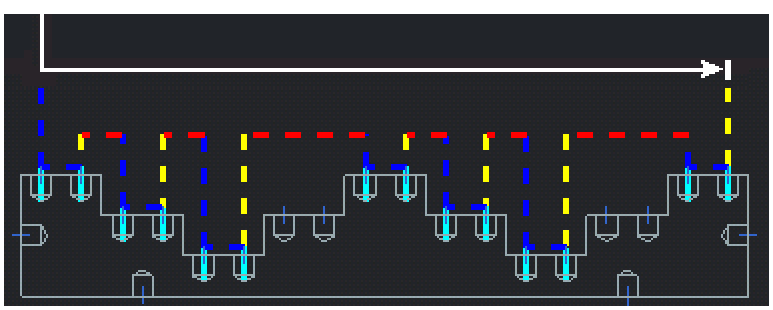
Direct



Clearance Only, 7 cycles



Clearance Initial, 2 cycles



Always, 1 cycle

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Operation **Type** = **hole_making**
Location in dialog box	[Drilling operation] dialog box→**Machine Control** group→**Retract Output Mode** list

Feature group enhancements (10.0.1)

You now have the same control when using an automatic feature group that you have in a manually defined **HOLE_BOSS_GEOM** geometry group. Operations within the feature group geometry parent hierarchy also use the sequence defined in the feature group. In the **Feature Group** dialog box, you can:

- Optimize the sequence of features within the group automatically, using one of the **Optimization** options.

- Reorder the sequence of features within the group manually.

  In the list, select one or more features and use the arrows to move the features as required.

- Reverse the machining direction of selected features.

When this dialog box is open, you can change the orientation of a single feature using the dynamic manipulator.

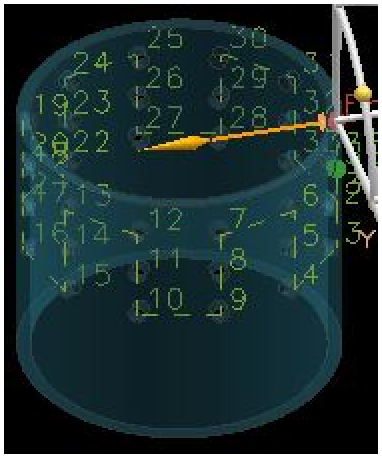
You can control the preview color of the feature sequence using the **Feature Sequence** customer default.

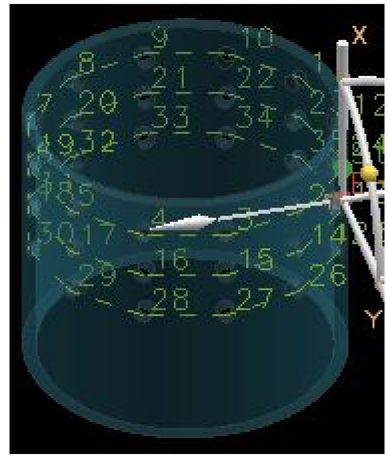
Tip

To find a customer default, choose **File**→**Utilities**→**Customer Defaults**, and click **Find Default** .

If needed, you can still access the legacy classification dialog box.

Optimize the feature sequence

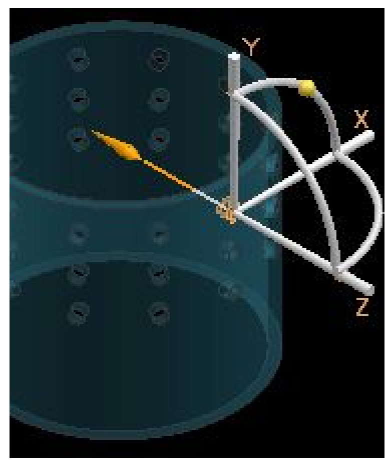




Closest

Primary Direction = X

Reverse machining direction

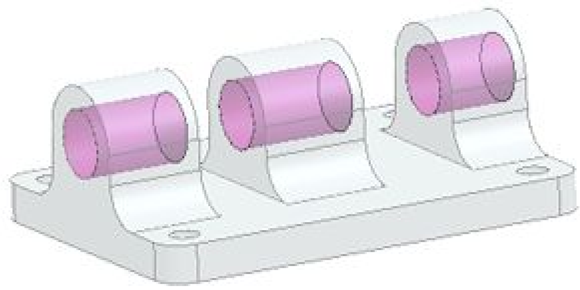


Where do I find it?

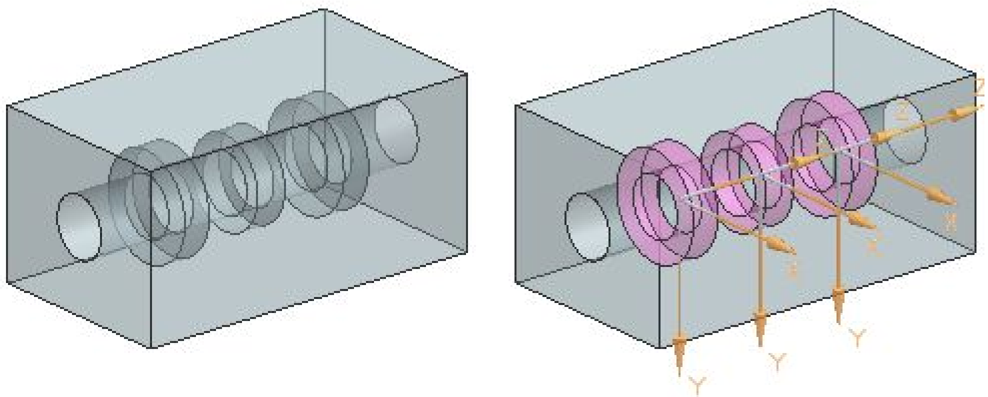
Initial release	NX 9.0.3 MP1
Application	Manufacturing
Operation Navigator	Double-click the feature group

Feature recognition enhancements (10.0.1)

NX now recognizes coaxial interrupted holes. The feature type is **HOLE_ROUND_INTERRUPTED_STRAIGHT**.



NX now recognizes radial grooves. The feature type is **GROOVE_INS_RAD_RECT**.



Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Machining Feature Navigator	Right-click in the background→**Find Features**

Allowed violations of part geometry (10.0.1)

In the path settings for an operation, when you check for gouges, NX normally reports a gouge and does not generate a gouging tool path.

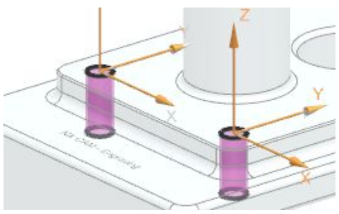
For some situations, you implicitly authorize a gouge to the modeled body by defining a machining feature that is not explicitly modeled. In these situations, NX generates a tool path and does not report gouges within a defined feature volume. For authorized gouges, NX gouge checking reports `No gouged motions found`.

The following are examples of implicitly authorized gouges.

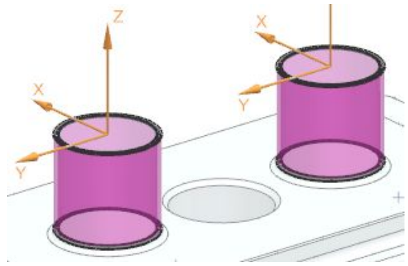
Tapping and thread milling

Typically, you model only the tap diameter of a hole or the maximum thread diameter of a boss. NX creates the feature volume during the machining feature recognition process.

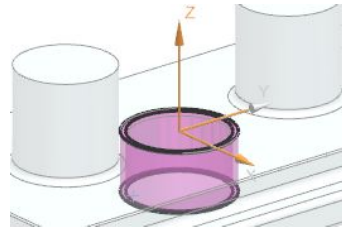
Tapped hole volumes



Thread mill boss volume



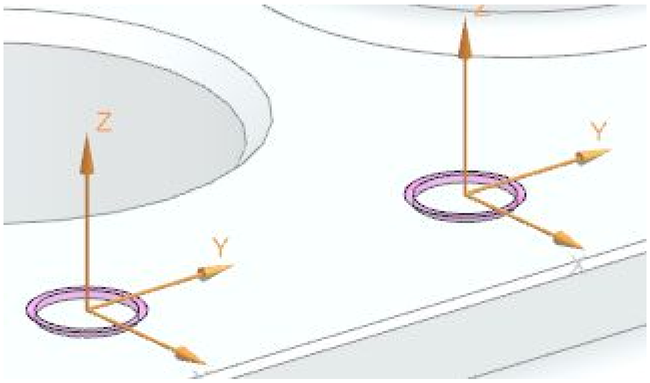
Thread mill hole volume



Chamfering holes

You need not model the chamfer.

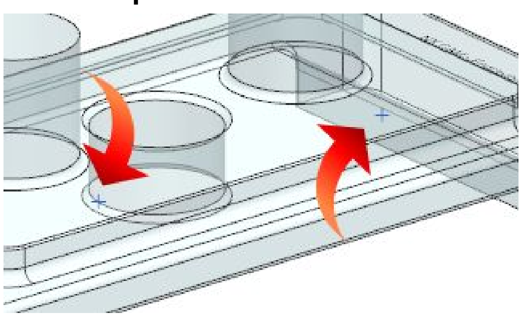
Chamfer volumes



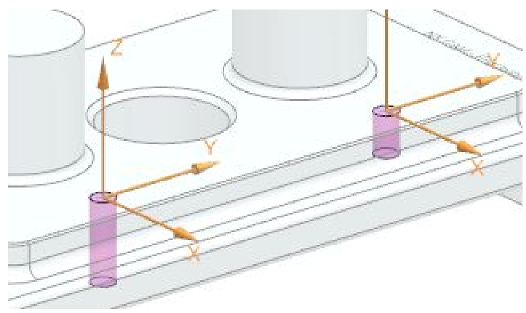
Drilling holes at points

You can specify a hole feature at a point, without modeling the hole.

Modeled points



Hole feature volumes

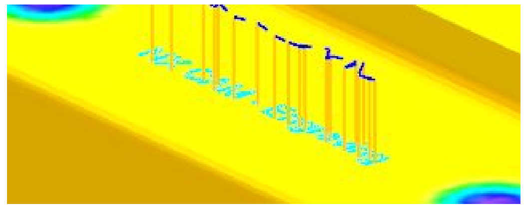


Engraving

When you engrave text using **Planar Text** and **Contour Text** operations, the text to be cut exists only as a 2D note. There is no corresponding feature volume, and the operation dialog box does not contain a gouge checking option. No gouge is reported when you run a gouge check.

2D drafting note

Tool path



Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Prerequisite	Feature volumes must be defined for the operation.
Operation Navigator	Right-click operation→**Gouge Check** Right-click operation→**Tool Path**→**Verify**→**Replay**→**Gouge and Collision Settings**
Operation dialog box	**Generate** **Verify**→**Replay**→**Gouge and Collision Settings**
Tool Path Editor dialog box	**Analyze Tool Path** group→**Gouge and Collision Check**
Simulation Control Panel dialog box	For tapping and thread milling only: **Simulation Settings** group→**Simulation Settings** →**Simulation Settings** dialog box→**Collision Detection** group→**Specify Collision Pairs** →**Specify Collision Pairs** dialog box→specify the part and the tool

Axis dynamic capabilities in the Machine Tool Navigator (10.0.1)

What is it?

Dynamic properties of axes are now stored in a common location, the NX machine tool assembly. You enter the properties from the **Machine Tool Navigator**, using the **Edit**→**Axis** command. The **Machine Tool Navigator** is available in the Manufacturing and Machine Tool Builder applications.

The following properties are new in NX.

For rotary axes and spindles	For linear axes
Max Acceleration [deg/s^2]	**Max Acceleration [mm/s^2]**
Max Deceleration [deg/s^2]	**Max Deceleration [mm/s^2]**
Jerk Limit [deg/s^3]	**Jerk Limit [mm/s^3]**
Jump Velocity [deg/s]	**Jump Velocity [mm/s]**
Kv	**Kv**
Fine Precision [deg]	**Fine Precision [mm]**
Coarse Precision [deg]	**Coarse Precision [mm]**

Units of mm for linear axes are shown. For English units, **in** replaces **mm**.

When you retrieve a machine tool model, NX checks whether machine tool axis and channel data has ever been modified or imported, and notifies you if the data has not been updated.

You can import axis dynamic and channel data from an MCF file or enter it manually. You will receive a one-time message that the link between the kinematic model and the MCF is broken, and that simulation will use the axis dynamic data in the part file.

Why should I use it?

- You do not have to enter the same information in multiple applications.
- You can enjoy the relative ease of using the Machine Tool Builder application, without having to learn the more advanced **Machine Configurator** dialog box in the common simulation environment (CSE).

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing Machine Tool Builder
Prerequisite	An open NX assembly that contains a machine tool kinematic structure
Machine Tool Navigator	Right-click the root node→**Import Axis and Channel Data** Right-click an axis node→**Edit**→**Axis**→**Dynamic Properties** group

Channels settings moved (10.0.1)

What is it?

You can now assign axes to channels in the **Machine Tool Navigator** in NX.

You can still assign a device or a tool holder to a channel to a in the **Edit Machine Component** dialog box; however, the options to rename, add, or delete a channel are consolidated in the **Channel Configuration** dialog box.

Why should I use it?

- You no longer have to use the **Machine Configurator** dialog box of the common simulation environment (CSE) to assign axes to channels.

Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing Machine Tool Builder
Prerequisite	An open NX assembly that contains a machine tool kinematic structure
Machine Tool Navigator	Right-click the root node → **Channel Configuration**

Toolpath verification and analysis consolidation (10.0.1)

What is it?

In previous releases, new functionality was added beside existing functionality. This resulted in some duplication and confusion over which command to use for a given task. Some older functions are now hidden by default in the **Tool Path Verification** dialog box.

2D Dynamic tab

The **2D Dynamic** tab is now hidden. This protects you from issues that might arise if, for example, a collision occurs but is not visible in the selected 2D view.

To restore the **2D Dynamic** tab, select the **Show 2D Dynamic Page** customer default.

Tip

To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

3D Dynamic tab

To ensure maximum reliability, on the **3D Dynamic** tab, the **Facetted Solid** options are hidden.

We recommend that you click **Show Thickness by Color** to check the amount of excess material that remains on the part.

We recommend that you use the following methods for gouge checking:

- Right-click an operation in the **Operation Navigator** and choose **Tool Path**→**Gouge Check**.

- In the **Tool Path Verification** dialog box, on the **Replay** tab, click **Gouge and Collision Settings**.

To restore the **Facetted Solid** options, select the **Enable Facetted Solid for Gouges and Excess** customer default.

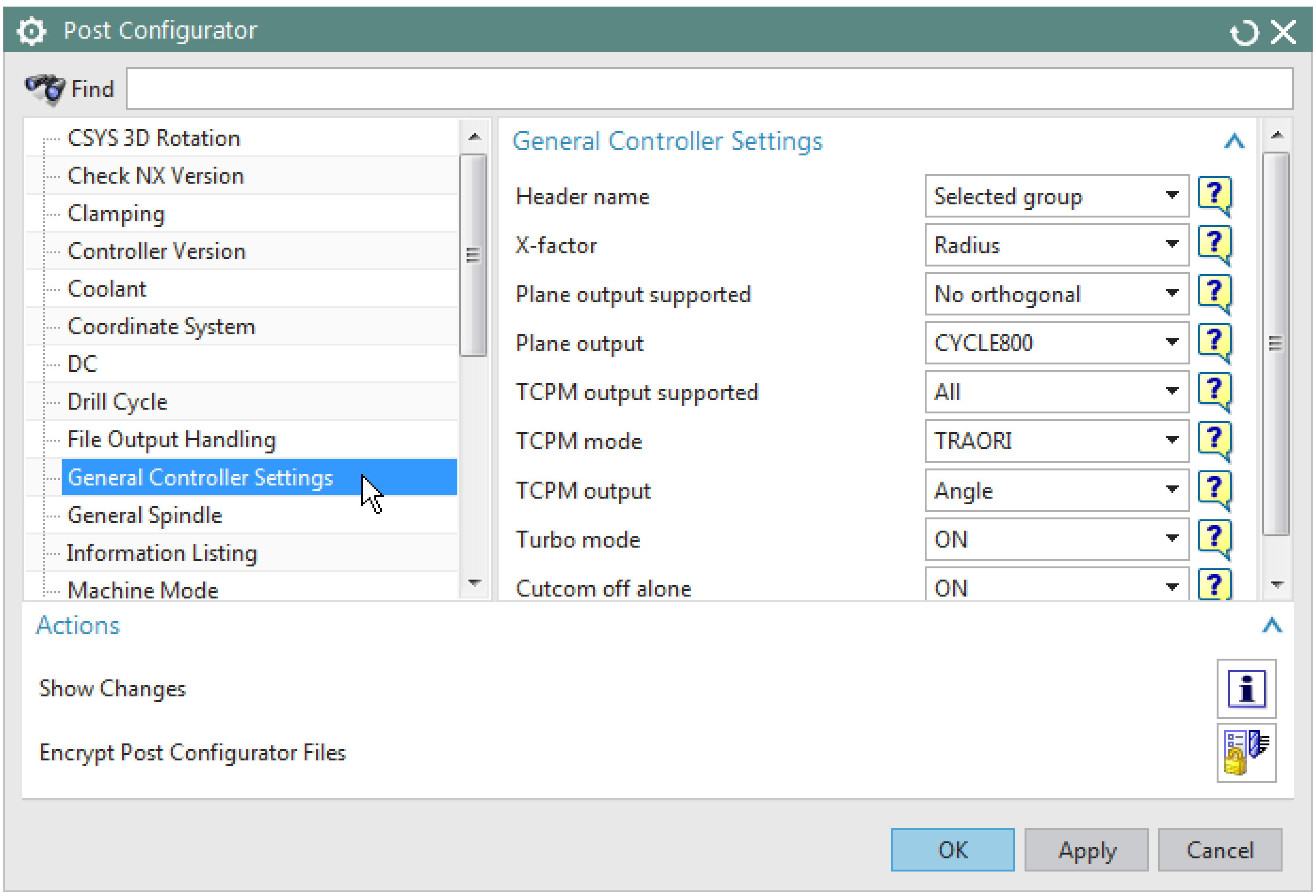
Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing

Post Configurator (10.0.1)

Use the **Post Configurator** to create postprocessors for the following controllers in an efficient way:

- Sinumerik 840D
- Fanuc 18i
- Heidenhain iTNC 530
- Generic



You can create a new post processor that is based on a loaded kinematic machine model. If the CAM setup does not contain a kinematic model, you must set it up manually.

Why should I use it?

The **Post Configurator** offers a new way of creating postprocessors based on the established manufacturing output manager (MOM) architecture. If you create post processors, you can use and configure the included post libraries. There are three stages of licenses available for NX CAM users.

Basic

This license is available for all NX CAM users. It allows you to:

- Run a postprocessor that was created with Post Configurator.
- Edit parameters to which access is granted by the post processor developer.

Advanced

The license is intended for machine tool manufacturers and field service technicians.

It is available for all NX CAM users who currently own a Post Builder license. In addition to do the tasks that you can do with the basic license you can:

- Create a new post processor.
- Edit your post processor using the Post Configurator dialog box.

Full

The post developer license.

The license must be purchased. It is designed for Siemens Professional Services staff, Siemens partners, and NX CAM users who are heavily involved in creating post processors. In addition to the tasks that you can do with the advanced license, you can also:

- Edit the Tcl code.
- Encrypt your post processors.
- Limit the running of a post processor to a specific Sold-To ID.
- Define an expiration date for your post processors.

Note

When you create post processors using the **Post Configurator**, you can edit them only in the **Post Configurator**. If you created post processors using the **Post Builder**, you will not be able to edit them in the **Post Configurator**.

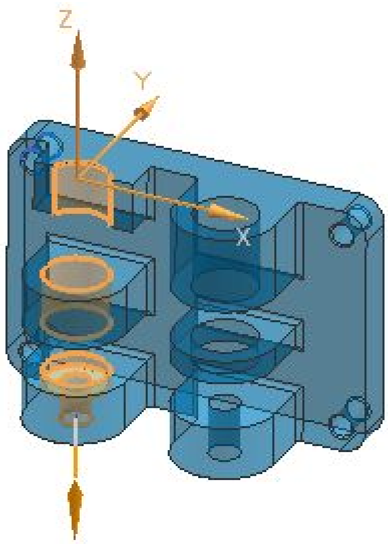
Where do I find it?

Application	Manufacturing
Ribbon	**Home** tab→**Tools** drop-down→**Post Configurator**
Menu	**Menu**→**Tools**→**Post Configurator**

Feature recognition for coaxial interrupted holes (10.0.1)

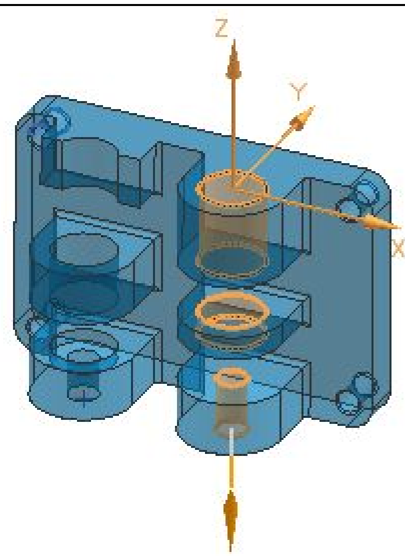
NX first tries to recognize a sequence of coaxial interrupted holes as the appropriate **STEPx** feature so that the machining process is the same for holes with and without interruptions. If NX does not recognize a sequence of coaxial interrupted holes as a regular stepped feature, it classifies the feature as **HOLE_ROUND_INTERRUPTED_STRAIGHT**. The machining process is different for these holes.

In the following examples, both features consist of coaxial cylinders that extend through multiple flanges.



STEP2HOLE

The lowest flange contains a **STEP2HOLE** feature. The additional cylinders that extend through the remaining flanges have the same diameter as the largest **STEP2HOLE** diameter.



HOLE_ROUND_INTERRUPTED_STRAIGHT

The lowest flange contains a **STEP1HOLE** feature. In this case, the diameters of the additional cylinders in the remaining flanges do not match the diameter of the **STEP1HOLE** feature. This hole requires a different machining process than a **STEP1HOLE** feature.

You can use both **STEPx** and **HOLE_ROUND_INTERRUPTED_STRAIGHT** features as input for the **Sequential Drilling** operation.

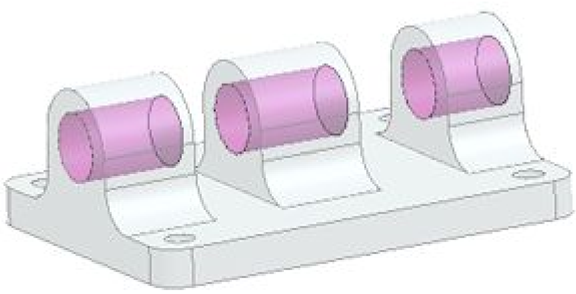
Where do I find it?

Initial release	NX 10.0.1
Application	Manufacturing

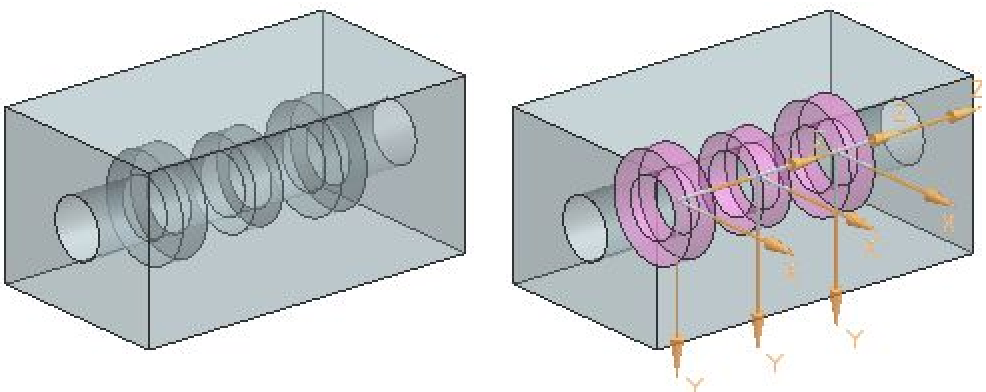
Machining Feature Navigator	Right-click in the background→**Find Features**

Feature recognition enhancements (10.0.1)

NX now recognizes coaxial interrupted holes. The feature type is **HOLE_ROUND_INTERRUPTED_STRAIGHT**.



NX now recognizes radial grooves. The feature type is **GROOVE_INS_RAD_RECT**.



Where do I find it?

Initial release	NX 9.0.3 MP1
Application	Manufacturing
Machining Feature Navigator	Right-click in the background→**Find Features**

Solver version support (10.0.1)

For each released version of NX, the following tables list the supported solver versions for import, export, and the post-processing of results. Note:

- The version listed in the **Import ASCII** and **Import Binary** rows is the solver version that was generally available when the NX version was released. In general, the import of the solver ASCII and binary files should be upwards compatible. Therefore, you should be able to import them into the most recent version of NX. However, in general:

- o ASCII files are backwards compatible for import into NX. If you import an ASCII file from a newer version of the solver than is officially supported, the software simply ignores any new fields/options that aren't supported in the current NX release.
- o Binary files are not backwards compatible. For example, you can import a binary file created by NX Nastran 5.0 into NX 6.0.2, but you might not be able to import a binary file created by NX Nastran 6.1 into NX 5.

- The version listed in the **Export ASCII** rows is the solver version that was available when the NX version was tested. In general, the exported solver input file is upwards compatible for that solver. Backwards compatibility is not guaranteed. For NX Nastran, the **Model Setup Check** function in Advanced Simulation tries to flag potential version incompatibility issues.
- The version listed in the **Post-processing Results** rows is the version of the solver results that was tested in the listed NX version. In general, results from earlier solver versions are also supported.

NX 10 releases

Solver	File Type	NX 10	NX 10.0.1
NX Nastran	Import ASCII (.dat)	10	10
	Import Binary (.op2)	10	10
	Export ASCII (.dat)	10	10
	Post-processing of Results (.op2, .xdb)	10	10.1
MSC Nastran	Import ASCII (.dat)	2013.1	2014
	Import Binary (.op2)	2013.1	2014
	Export ASCII (.dat)	2013.1	2014
	Post-processing of Results (.op2, .xdb)	2013.1	2014
Abaqus	Import ASCII (.inp)	6.13	6.13
	Import Binary	N/A	N/A
	Export ASCII (.inp)	6.13	6.13
	Post-processing of Results (.fil)	6.14	6.14-1
	Post-processing of Results (.odb)	6.13-5	6.13-5
ANSYS	Import ASCII (PREP7, CDB)	15.0	15.0
	Import Binary (.rst, .rth)	15.0	15.0
	Export ASCII (.inp)	15.0	15.0
	Post-processing of Results	15.0	15.0
LS-DYNA	Import ASCII	971R6.0	971R6.0
	Import Binary	N/A	N/A
	Export ASCII (.k)	971R6.0	971R6.0
	Post-processing of Results	971R7.1	971R7.1
Permas	Post-processing of Results (.res, .post)	13	13

Solver	File Type	NX 10	NX 10.0.1
SAMCEF	Import ASCII (.dat)	16.1 (build 16.1-02)	16.1 (build 16.1-02)
	Import Binary (.sdb)	16.1 (build 16.1-02)	16.1 (build 16.1-02)
	Export ASCII (.dat)	16.1 (build 16.1-02)	16.1 (build 16.1-02)
	Post-processing of Results (.des, .fac)	16.1 (build 16.1-02)	16.1 (build 16.1-02)

NX 9 releases

Solver	File Type	NX 9	NX 9.0.1	NX 9.0.2	NX 9.03
NX Nastran	Import ASCII (.dat)	9	9	10	10
	Import Binary (.op2)	9	9	10	10
	Export ASCII (.dat)	9	9	10	10
	Post-processing of Results (.op2)	9	9	10	10
MSC Nastran	Import ASCII (.dat)	2013	2013	2013.1	2013.1
	Import Binary (.op2)	2013	2013	2013.1	2013.1
	Export ASCII (.dat)	2013	2013	2013.1	2013.1
	Post-processing of Results (.op2)	2013	2013	2013.1	2013.1
Abaqus	Import ASCII (.inp)	6.12	6.12	6.12	6.12
	Import Binary	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.12	6.12	6.12	6.12
	Post-processing of Results (.fil)	6.12-1	6.12-1	6.13-4	6.13-4
	Post-processing of Results (.odb)	6.12	6.12	6.12	6.13-2
ANSYS	Import ASCII (PREP7, CDB)	14.5	14.5	15.0	15.0
	Import Binary (.rst, .rth)	14.5	14.5	15.0	15.0
	Export ASCII (.inp)	14.5	14.5	15.0	15.0
	Post-processing of Results	14.5	14.5	15.0	15.0
LS-DYNA	Import ASCII	971R6.0	971R6.0	971R6.0	971R6.0
	Import Binary	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R6.0	971R6.0	971R6.0	971R6.0
	Post-processing of Results	971R6.0	971R6.0	971R7.1	971R7.1
Permas	Post-processing of Results (.res)	13	13	13	13

NX 8 releases

Solver	File Type	NX 8	NX 8.0.1	NX 8.0.2	NX 8.0.3	NX 8.5	NX 8.5.1	NX 8.5.2
NX Nastran	Import ASCII (.dat)	8	8	8.5	8.5	8.5	8.5	8.5
	Import Binary (.op2)	8	8	8.5	8.5	8.5	8.5	8.5
	Export ASCII (.dat)	8	8	8.5	8.5	8.5	8.5	8.5
	Post-processing of Results (.op2)	8	8.1	8.5	8.5	8.5	8.5	9
MSC Nastran	Import ASCII (.dat)	2011.1	2011.1	2012.1	2012.1	2012.1	2012.1	2012.1
	Import Binary (.op2)	2011.1	2011.1	2012.1	2012.1	2012.1	2012.1	2012.1
	Export ASCII (.dat)	2011.1	2011.1	2012.1	2012.1	2012.1	2012.1	2012.1
	Post-processing of Results (.op2)	2011.1	2011.1	2012.1	2012.1	2012.1	2012.1	2013
Abaqus	Import ASCII (.inp)	6.10	6.10	6.10	6.10	6.12	6.12	6.12
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.10	6.10	6.10	6.10	6.12	6.12	6.12
	Post-processing of Results (.fil)	6.11	6.11	6.11	6.12-1	6.12-1	6.12-1	6.12-1
	Post-processing of Results (.odb)	6.10-EF1	6.11	6.11	6.11	6.12	6.12	6.12
ANSYS	Import ASCII (PREP7, CDB)	13	13	14	14	14	14	14.5
	Import Binary (.rst, .rth)	13	13	14	14	14	14	14.5
	Export ASCII (.inp)	13	13	14	14	14	14	14
	Post-processing of Results	13	13	14	14	14	14	14
LS-DYNA	Import ASCII	971R5.0	971R5.0	971R5.0	971R5.0	971R6.0	971R6.0	971R6.0
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R5.0	971R5.0	971R5.0	971R5.0	971R6.0	971R6.0	971R6.0
	Post-processing of Results	971R5.0	971R5.0	971R5.0	971R5.0	971R6.0	971R6.0	971R6.0

NX7 releases

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2	NX 7.5.3	NX 7.5.4	NX 7.5.5.
NX Nastran	Import ASCII (.dat)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Import Binary (.op2)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Export ASCII (.dat)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Post-processing of Results	6.1	7.0	7.1	7.1	7.1	7.1	8
MSC Nastran	Import ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Import Binary (.op2)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Export ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Post-processing of Results	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
Abaqus	Import ASCII (.inp)	6.8-1	6.9–1	6.9–1	6.9-1	6.10	6.10	6.10
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9	6.9	6.9	6.10	6.10	6.10
	Post-processing of Results (.fil)	6.8-EF2	6.9.2	6.9.2	6.10-1	6.10-1	6.10-1	6.11-1
	Post-processing of Results (.odb)	6.8-EF2	6.9-EF1	6.9-EF2	6.9-EF2	6.10-EF1	6.10-EF1	6.10-EF1
ANSYS	Import ASCII (PREP7, CDB)	12	12.1	12.1	12.1	13	13	13
	Import Binary (.rst, .rth)	12	12.1	12.1	12.1	13	13	13
	Export ASCII (.inp)	12	12.1	12.1	12.1	13	13	13
	Post-processing of Results	12	12.1	12.1	12.1	12.1	12.1	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1

NX 6 releases

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3	NX 6.0.4	NX 6.0.5
NX Nastran	Import ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Import Binary (.op2)	6.0	6.1	6.1	6.1	6.1	7.0
	Export ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Post-processing of Results	6.0	6.0	6.1	6.1	7.0	7.0
MSC Nastran	Import ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Import Binary (.op2)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Post-processing of Results	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
Abaqus	Import ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Post-processing of Results (.fil)	6.7-5	6.8-1	6.8-3	6.8-EF2	6.8-EF2	6.8-EF2
	Post-processing of Results (.odb)	N/A	N/A	N/A	6.8-EF	6.8-EF2	6.9-EF2
ANSYS	Import ASCII (PREP7, CDB)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Import Binary (.rst, .rth)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Export ASCII (.inp)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Post-processing of Results	11 SP1	11 SP1	11 SP1	11 SP1	12.0	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R2	971R2	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A	N/A	N/A	N/A	N/A

NX 5 releases

Solver	File Type	NX 5	NX 5.0.1	NX 5.0.2	NX 5.0.3	NX 5.0.4	NX 5.0.5	NX 5.0.6
NX Nastran	Import ASCII (.dat)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Import Binary (.op2)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Export ASCII (.dat)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Post-processing of Results	5.0	5.0	5.1	5.1	5.1	5.1	6.0
MSC Nastran	Import ASCII (.dat)	2005	2005	2007	2007	2007	2007	2007r1
	Import Binary (.op2)	2005	2005	2007	2007	2007	2007	2007r1
	Export ASCII (.dat)	2005	2005	2007	2007	2007	2007	2007r1
	Post-processing of Results	2005	2005	2007	2007	2007	2007	2008r1
Abaqus	Import ASCII (.inp)	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.7-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.7-1
	Post-processing of Results	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.8-1
ANSYS	Import ASCII (PREP7, CDB)	10	10	11	11	11	11	11
	Import Binary (.rst, .rth)	10	10	11	11	11	11	11
	Export ASCII (.inp)	10	10	11	11	11	11	11
	Post-processing of Results	10	11	11	11	11	11	11 SP1

NX 4 releases

Solver	File Type	NX 4	NX 4.0.1	NX 4.0.2	NX 4.0.3	NX 4.0.4
NX Nastran	Import ASCII (.dat)	4.0	4.1	4.1	5.0	5.0
	Import Binary (.op2)	4.0	4.1	4.1	4.1	4.1
	Export ASCII (.dat)	4.0	4.1	4.1	5.0	5.0
	Post-processing of Results	4.0	4.1	4.1	5.0	5.0
MSC Nastran	Import ASCII (.dat)	2005	2005	2005	2005	2005
	Import Binary (.op2)	2005	2005	2005	2005	2005
	Export ASCII (.dat)	2005	2005	2005	2005	2005
	Post-processing of Results	2005	2005	2005	2005	2005

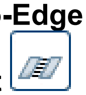
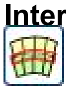
Solver	File Type	NX 4	NX 4.0.1	NX 4.0.2	NX 4.0.3	NX 4.0.4
Abaqus	Import ASCII (.inp)	6.5-1	6.5-1	6.5-1	6.6	6.6
	Import Binary	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.5-1	6.5-1	6.5-1	6.6	6.6
	Post-processing of Results	6.5-1	6.5-1	6.5-1	6.6	6.6-3
ANSYS	Import ASCII (PREP7, CDB)	8	9	9	10	10
	Import Binary (.rst, .rth)	8	9	9	10	10
	Export ASCII (.inp)	8	9	9	10	10
	Post-processing of Results	9	9	9	10	10

Abaqus axisymmetric contact analysis (10.0.1)

What is it?

This release includes expanded support for contact analyses in the Abaqus environment. You can now perform a number of different types of contact analyses in axisymmetric structural and thermal models. In previous releases, contact analyses in the Abaqus environment were limited to non-axisymmetric models.

The following table summarizes the commands that are now available for axisymmetric structural and thermal solutions:

Abaqus element	Description	Analysis types supported	Corresponding Abaqus keywords
Edge-to-Edge Contact 	Defines contact conditions between selected polygon edges or element edges.	Axisymmetric structural	`*CONTACT PAIR` `*SURFACE`
Contact Interference 	Defines time-dependent allowable interferences for contact pairs and elements. You can use **Contact Interference** to model interference fits (press fits) when there are large overclosures between the contacting surfaces in the initial configuration of the model. The defined contact interferences can result in stresses and strains as the solver resolves the overclosure	Axisymmetric structural	`*CONTACT PAIR` `*CONTACT INTERFERENCE`

Abaqus element	Description	Analysis types supported	Corresponding Abaqus keywords
	between the contacting surfaces.		
Contact with Clearance	Defines precise clearance or overclosure values for the nodes on the slave (target) surface in a contact pair. An overclosure value allows you to specify any initial penetration for the contacting surfaces.	Axisymmetric structural	`*CONTACT PAIR` `*CLEARANCE`
Surface-to-Surface Thermal Conductance	Models conductive heat transfer between proximate or contacting surfaces. You can model the conductive heat transfer as a function of: • The clearance between the contacting surfaces. • The contact pressure at the interface between the contacting surfaces. • Both the clearance and the contact pressure.	Axisymmetric thermal	`*GAP CONDUCTANCE`

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus as the specified solver Axisymmetric Structural or Axisymmetric Thermal as the analysis type General Analysis or Heat Transfer as the solution type

ANSYS axisymmetric contact analysis (10.0.1)

What is it?

This release includes support for contact analyses in ANSYS axisymmetric structural and thermal solutions. In previous releases, contact was supported only in non-axisymmetric solutions.

Axisymmetric structural solutions

You can now use the **Structural Contact** command to analyze contact between the edges of planar elements in axisymmetric models. You use the **Structural Contact** and the **Region** dialog boxes to define contact and sliding between a 2D target edge and a deformable surface. When you export or solve the solution, the software creates the appropriate CONTA172 elements to define the deformable surface and TARGE169 elements to define the target surface.

Axisymmetric thermal solutions

You can now use the **Thermal Contact** command to define thermal contact conduction between contacting surfaces in axisymmetric models using the edges of planar elements. You use the **Thermal Contact** and the **Region** dialog boxes to define the contact between the edges, as well as select the edges, points, or nodes that define the contacting edges.

After you define the contacting surfaces, use the **CONTA174/172 Real Constants** dialog box to define the required thermal contact properties. You can define the following properties:

- **Thermal contact conductance** (`TCC`), which allows you to take into account the conductive heat transfer between the contact and target surfaces.

  Note

  Currently, NX supports only a constant **Thermal contact conductance** value. You cannot vary the **Thermal contact conductance** with temperature, pressure, or time.

- The **Stefan-Boltzmann constant** (`SBCT`), which you can use to model radiative heat transfer. If you do not specify a value for the **Stefan-Boltzmann constant**, the radiation effect is excluded from the thermal contact calculations.

- The **Radiation view factor** (`RDVF`), which you can use to define the radiation view factor for near-field radiation.

Where do I find it?

Structural Contact

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Axisymmetric Structural as the analysis type Linear Statics or Nonlinear Statics as the solution type
Command Finder	**Structural Contact**

Thermal Contact

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part ANSYS as the specified solver Axisymmetric Thermal as the analysis type Steady State Thermal as the solution type
Command Finder	**Thermal Contact**

Constraint equation support in Abaqus and ANSYS thermal solutions (10.0.1)

What is it?

You can now create constraint equations between the temperature degrees-of-freedom of thermal elements in both the Abaqus and ANSYS thermal environments. In previous releases, you could only create constraint equations in the Abaqus and ANSYS structural environments.

Constraint equations for Abaqus thermal solutions

In the Abaqus thermal environment, use the new **Manual Coupling** command to define the independent and dependent coupling parameters between selected nodes. When you export or solve your model, NX creates the appropriate `*EQUATION` commands in your Abaqus input file.

Constraint equations for ANSYS thermal solutions

In the ANSYS thermal environment, the **Manual Thermal Coupling** command has been enhanced to allow you to create constraint equations as well as define sets of coupled degrees-of-freedom. Use the new **CE** option in the **Type** list of the **Manual Thermal Coupling** dialog box to define coupling between the temperature degrees-of-freedom within the model. When you export or solve your model, the software creates the appropriate `CE` keywords in your ANSYS input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file as the displayed part and the work part Abaqus or ANSYS as the specified solver Thermal as the analysis type
Command Finder	**Manual Coupling** (Abaqus) **Manual Thermal Coupling** (ANSYS)

CGNS results file options (10.0.1)

What is it?

You can now request transient flow results in the CGNS format to be written in one of the following ways:

- One CGNS results file: The results file contains results from all transient output times.
- Multiple CGNS results files: Each results file contains results from one transient output time.

In previous releases, NX always wrote a single CGNS results file containing results from all transient output times.

Why should I use it?

You can now obtain the results from each transient output time in its own CGNS results file. You do this by requesting multiple CGNS files. You can continue to request a single CGNS file that contains all the results from all transient output times.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Command Finder	**Solution**
Simulation Navigator	Right-click a solution node→**Edit**
Location in dialog box	**Results Options** page→**Optional Output Format** group→**CGNS** →**Transient Results** list

Inflated laminates in LS-DYNA (10.0.1)

What is it?

The **Extrude Laminate** and **Fill Laminate** commands are now supported in the LS-DYNA environment.

The extruded 3D mesh consists of ELEMENT_TSHELL wedge or hexahedral elements and the ply drop-offs consist of ELEMENT_SOLID tetrahedral, wedge, or hexahedral elements. Because LS_DYNA does not support pyramids, pyramids are split into tetrahedral elements.

For both commands, NX creates a 3D mesh collector that has a **Solid Laminate** physical property assigned to it. NX automatically sets the **Stacking Recipe** in the **Solid Laminate Modeler** dialog box to **Inherited from Layup**.

All the plies of the extruded solid laminate have the same ply failure theory and inter-laminar failure theory that you selected in the **Laminate Modeler** dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM within the LS-DYNA solver environment.
Command Finder	**Extrude Laminate**  or **Fill Laminate** 

MCOSMOS postprocessor (10.0.1)

What is it?

MCOSMOS, the Mitutoyo Controlled Open Systems for Modular Operation Support postprocessor, is now available as part of the standard installation for NX 10.0.1. It is not necessary to download the files from GTAC.

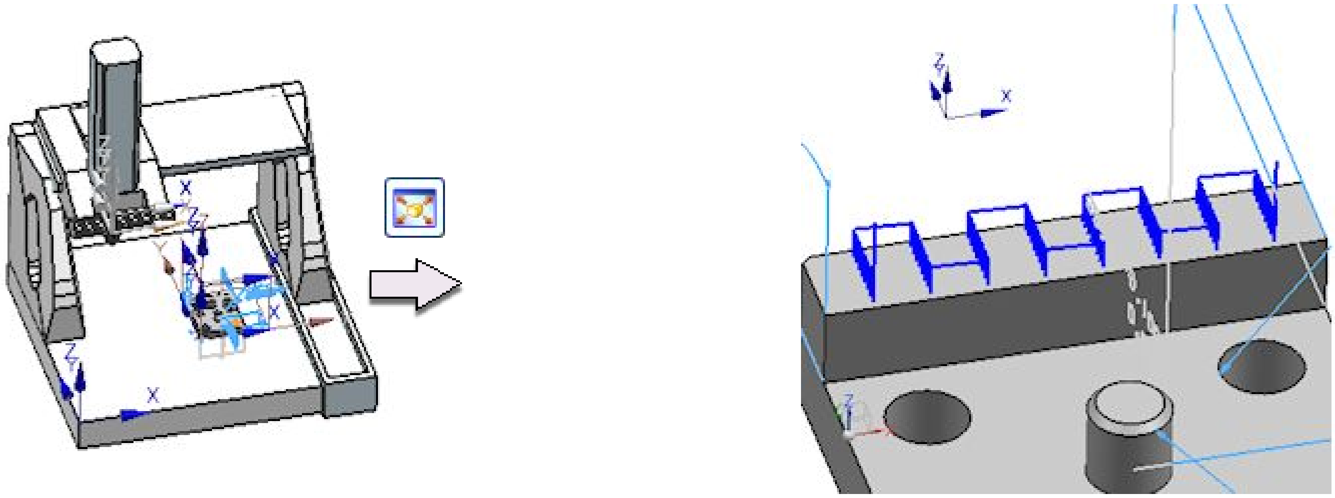
Where do I find it?

Application	CMM Inspection Programming
Command Finder	Postprocess
Location in dialog box	**Postprocess** dialog box→**Postprocessor** group→**MITUTOYO MCOSMOS**

CMM general enhancements (10.0.1)

What is it?

You can fit one or more selected objects to the graphics window view.



Where do I find it?

Application	CMM Inspection Programming
Prerequisite	Select the objects in the **Inspection Navigator**.
Top Border bar	**View** tab→**Orientation** group→**Fit View to Selection**

Converting legacy welds (10.0.1)

When you run the **Convert Legacy Weld Points** weld utility, you can specify if you want to simultaneously convert all or selected pre-NX 10 weld point features. To do this, use the **All in Work Part** option or the **Selected Only** option.

Where do I find it?

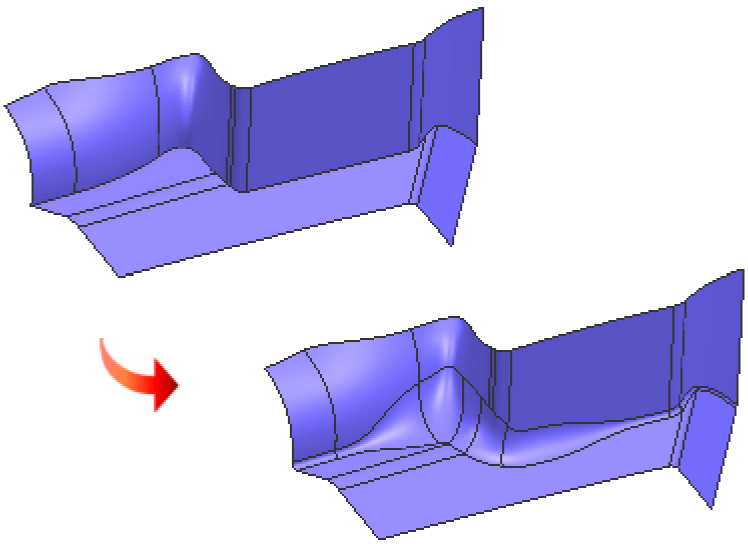
Application	Modeling
Prerequisite	Set the **Point Locator** customer defaults similar to the **Legacy Welds (pre NX10)** customer defaults. To select the **Selection Scope** options, you must select the **Convert** check box.
Command Finder	**Weld Utilities**
Location in dialog box	**Convert Legacy Weld Points** group→**Convert** check box **Selection Scope** subgroup→**All in Work Part** or **Selected Only**

Flow Blend (10.0.1)

Use the **Flow Blend** command to create constant or variable blends along complex faces.

You can:

- Blend complex faces that have a small curvature.
- Patch the blended faces to the original body.
- Smoothen variable blends.



Where do I find it?

Application	OmniCAD for NX
Prerequisite	To use OmniCAD commands, you must run the `InstallSentinel.bat` file which is available in the *ugii/omnicad* directory.
Command Finder	**Flow Blend**

Exporting crosshatching as grouped geometry (10.0.1)

What is it?

You can now export crosshatching as grouped geometry to a 2D NX Part, DXF, or DWG file. To do this, you must set the following keyword in the settings file.

```
CONVERT_CROSSHATCH_AS_GEOMETRY = yes
```

Note

This keyword setting overrides the default crosshatch mapping defined in the DXF/DWG and 2D Exchange translators.

Why should I use it?

When the crosshatch pattern assigned to the crosshatching in the NX file is not available in AutoCAD, the crosshatching display in AutoCAD may not match the display in NX. In this case, export the crosshatching as grouped geometry to get the exact display in AutoCAD.

Where do I find it?

Command Finder	**Export 2D Exchange File** **Export AutoCAD DXF/DWG File**

Line Designer enhancements in NX 10.0.1

- Added support for Drag & Drop of resources from History Navigator and Teamcenter Navigator to the graphics window.
- Improved mouse button menus.
  - o Removed commands that are not applicable in Line Designer 4GD.
  - o Removed low-use commands; rearranged order for frequent access; added high-use commands.
  - o Streamlined Smart Component shortcut menus.
- Added new command for disconnecting connected components.
- Enhanced Show/Hide and reference sets so that their state is saved with the workset.
- For 4GD mode, added the **Replace Source Part** command to the Plant Navigator context menu for resource instances. In BVR mode, the Replace Component command is applicable.
- Added an option in the connector definition for a robot that reverses the direction of propagation. If the source connector has this option enabled, then dropping the robot onto the connector of an existing instance of a riser causes the diameter from the robot to be propagated to the riser.
- Added **Type** and **Effectivity** columns to Plant Navigator. Note: In BVR mode, values in Plant Navigator columns only appear if the column is enabled in the Assembly Navigator.
- Provided a script to install Line Designer template parts to Teamcenter.

Fast placement coordinates pop-up (10.0.1)

What is it?

The **Fast Placement** capability in Line Designer has been enhanced:

- Added Z-coordinate to the coordinate pop-up.
- Added capability to edit X, Y, Z coordinate values in the coordinate pop-up.

Why should I use it?

You can use the coordinate display and editing capability to more precisely place components even while in the **Fast Placement** mode.

Where do I find it?

Application	Line Designer
Prerequisite	**Fast Placement** mode must be enabled.

Disconnect Components (10.0.1)

What is it?

The **Disconnect Components** command removes a connection between components that was created with the **Connect Components** command.

- If assembly constraints between the components exist, the assembly constraints are removed.
- Position and expression values of the components are not changed.

Why should I use it?

Disconnect components if you want to replace one of the components or change its location relative to the other component.

Where do I find it?

Application	Line Designer
Command Finder	**Disconnect Components** 

Drag and drop resources (10.0.1)

What is it?

In addition to adding resources to a layout from the Reuse Library, you can now drag and drop resources from the History Navigator and Teamcenter Navigator to the graphics window to create a new instance.

Why should I use it?

With this capability, you can take advantage of ready accessibility to existing resource instances when adding similar instances.

Where do I find it?

Application	Line Designer

Navigator	Resource instances listed in the **History Navigator** and **History Navigator**.

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