What's New in NX 8

Proprietary & Restricted Rights Notice

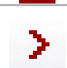
Contents

Chapter

1 *NX Help usability enhancements*

What is it?

NX Help has a new interface that makes it easier for you to find information and browse content.

	Returns you to the Home page.
	Displays the table of contents.
	Lets you navigate back through the hierarchy of Help categories.
	Displays search results. The Search box is available on every page.
	Displays search tips.
	Prints the current page.
	Displays links to related and supporting information.
	Plays video animations. Voice instructions are included.
	Lets you send feedback on content.

Click **Preferences** to change your search preferences. Enable **Filter search areas** to search in just the areas of NX that you use.

Why should I use it?

- Search performance is improved. You do not have to click a button to display the Search box. It is available at the top of the Help window. This makes search easier to access and decreases search time. When you click on a search results link, the page is displayed in the same browser window to reduce window clutter. To return to the search results page, use your browser's **Back** button or click **Search** again. Search filtering gives you control to search in specific areas of NX.

- Help is reorganized to make navigation more logical, to increase the relevancy of search results, and to make search filtering more specific.

- Breadcrumb links are available on each page to help you keep track of your location. A table of contents is still available, but is not displayed by default.

- Links to all related information are displayed on the right side. You no longer have to click the Options, How to, and other links on a page to see related information. Links are organized by information type under different headings.

Where do I find it?

Menu	**Help ® NX Help**
Keyboard accelerator	F1

Chapter

2 *HD3D*

Visual Reporting

Visual Reporting usability enhancements

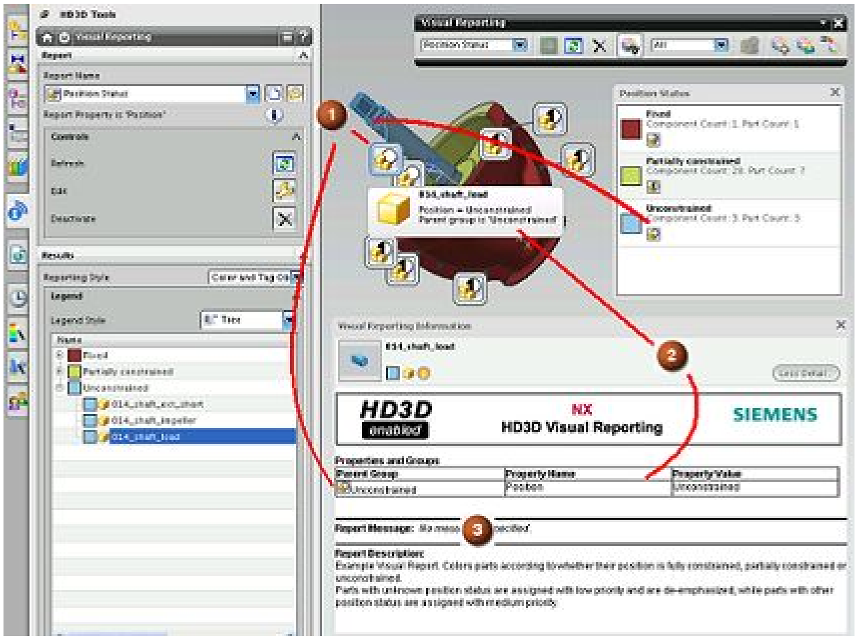
What is it?

- A new report selection command on the **Visual Reporting** toolbar allows you to quickly run visual reports from the toolbar at any time.

  The **Explore Visual Report** toolbar is renamed the **Visual Reporting** toolbar.

- Display of visual reports is enhanced.
  - You can define group priority to focus visibility in case of overlapping tags.
  - New layout template files define the default contents and layout of tool tips and of the **Visual Reporting Information** (Info View) dialog box. You can modify the layout template files to change the defaults. You can copy, modify, and then save template files with a new name in order to reference the modified templates for individual report formats.

- o You can define custom tag icons, custom tool tips, and custom result detail dialog boxes.



1 The bitmap image for the visual reporting tag.

2 A set of property information to include in either or both of the following:

- Visual reporting tag tool tips
- **Visual Reporting Information** (Info View) detail dialog boxes

3 A message string that appears in the additional information area of the **Visual Reporting Information** (Info View) dialog box.

- You can filter lists of part attribute names, lists of Teamcenter property names, and lists of user-specified entries in visual reporting dialog boxes by typing in an initial portion to match.
- You can send a report group or component part files to the Check-Mate or Requirements Validation tool.

Why should I use it?

You can select, activate, and deactivate HD3D Visual Reports using minimum options. You can make tags more distinctive according to report type and choose the information you want to display in report results.

Where do I find it?

Toolbar	**Visual Reporting**
Resource bar	**HD3D Tools® Visual Reporting® Edit**
Location in dialog box	**Visual Report Definition® Results** group
File system	*%UGII_BASE_DIR%\UGII\visual_reports\customization\templates\[language]*

Visual Reporting functionality enhancements

What is it?

- New capability to select multiple report definitions at once in the dialog box that appears when you click **Open Report**, or to define reports that contain multiple top-level properties. While the reports are active, you can switch among the top-level report properties in order to group and sort reported data in different ways.

  Use multiple-property reports to quickly group and sort data, and color objects, based on different report properties without having to sequentially run several different Visual Reports.

- You can compare properties in a scope term of a report definition.

- You can report on dates in report definitions.

- Units are displayed along with values in report results. By default, Visual Reporting now uses the top level assembly part unit system in conditions or grouping. You can specify a different unit system in the visual reporting definition file.

- The part attribute type “null“ reports on parts and components for which an attribute of a given name is defined, or on parts that do not include an attribute definition of a given name, without regard to the attribute’s value or type within the part or component.

- Existing reports are updated to utilize the new features available in NX 8.0.

- New properties in queries and new visual reports.

New properties available for visual reporting	New visual reports
o Check-Mate results	o Check-Mate results
o Check-Mate rule evaluation result	o Check-Mate rule evaluation result
o Requirements Validation status	o Requirements Validation status
o Last modified time	o Representation status
o Last modified user	o Parts modified since a given date time
o Parts modified in Teamcenter since the start of the current session	o Parts modified in Teamcenter since the start of the current session
o Number of freedom degrees of a component at its position related to Assembly constraints	o Parts created for a particular program
o Representation status	o Status of parts currently in a workflow[†]
	o Status of parts currently in a change process[†]
	o Parts affected by a change order[†]
	o Parts that have higher revisions than the ones that are being used in this assembly[†]
	[†]These reports use Teamcenter server side processing. Requires Teamcenter 9.

- New support for results of Teamcenter Report Builder (server-side) reports.

  Teamcenter 9 includes the following Report Builder (server side) reports for use with Visual Reporting:

  - Affected By Change Notice
  - Affected by Change Request
  - Affected by Problem Report
  - Has Higher Revision
  - Overall Status In Change Process
  - Overall Status In Workflow

  You can create additional server-side reports with the Teamcenter 9 Report Builder based on functions included with Teamcenter 9, or on custom functions written to gather other Teamcenter data of interest.

- By default, Visual Reporting now uses case insensitive comparison in conditions or grouping.

  You can specify case sensitive comparison in the visual reporting definition file.

- The **ugmgr_vpximport** utility for importing reports to Teamcenter has been enhanced with new switches for improved usability.

- A new **teamcenter_sample_reports_setup.pl** perl script has been added to the **ugmanager** folder for use in automatically importing into a Teamcenter database the out-of-the-box report definition files included with NX kits. The script includes an option to automatically create and set the Teamcenter preference **TC_NX_default_visual_report_folder_path** required for Visual Reporting.

Where do I find it?

Prerequisite	Teamcenter 9 is required for all server side reports. Server side reports related to change management require that the change management template is active in Teamcenter.
Resource bar	**HD3D Tools® Visual Reporting® Open Visual Report**
Location in dialog box	**Visual Report Definition® Report Property** group
Toolbar	**Visual Reporting**
Context menu	While a multi-property visual report is active, in the Visual Reporting Legend right-click a top-level (key) category® **Make Active Category**.
File system	*%UGII_BASE_DIR%\ugii\visual_reports\definitions\[language]* *%UGII_BASE_DIR%\ugmanager*

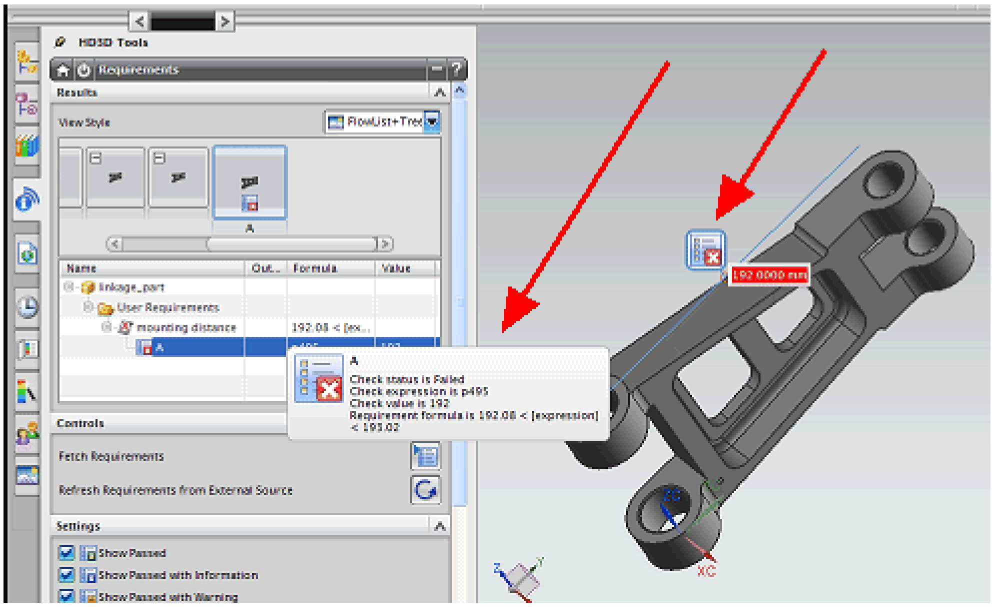
Requirements Validation

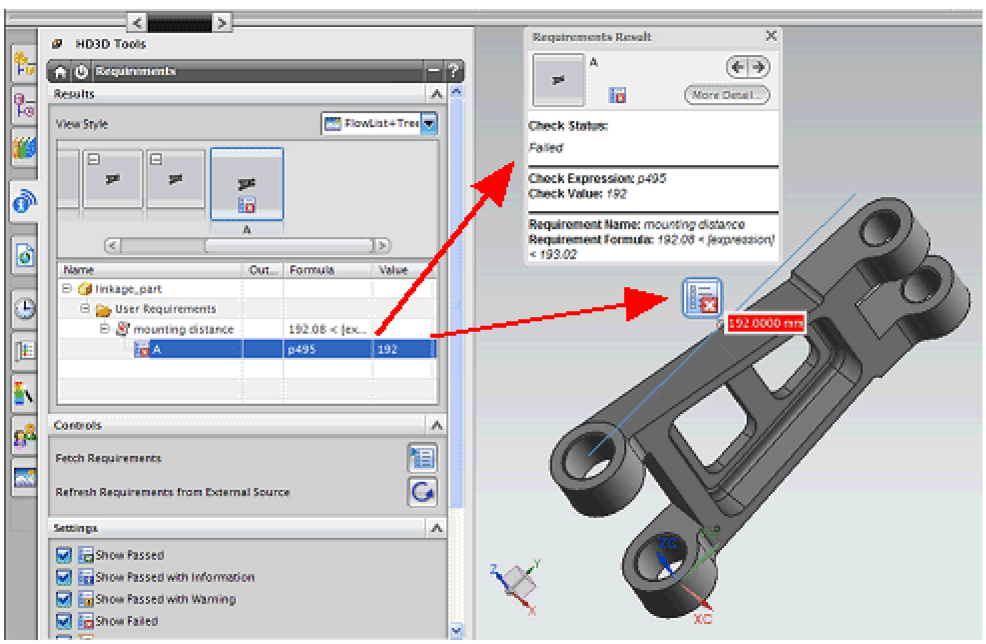
Requirements Validation in HD3D

What is it?

Use the Requirements Validation HD3D tool to create and import requirements, create requirement checks, and locate checks in the model geometry in order to check a design's compliance with requirements. New visual tags and reports in the HD3D tool and the graphics window provide easy access to detailed information.

Note The **Requirements Validation** command and HD3D tool replace and provide enhancements to functionality that in earlier releases was provided by a **Check Requirements** command and dialog box.





Where do I find it?

Menu	**Analysis® Requirements Validation**
Resource bar	**HD3D Tools® Requirements Validation**

HD3D tags in product templates

What is it?

As the result of this enhancement, you can now add HD3D tags to a product template to do the following:

- Provide a link to documentation for the template.
- Provide usage guidance for the template.
- Highlight failures or warnings generated by requirement checks in the template.
- Document and guide WAVE re-parenting operations in the template.

Where do I find it?

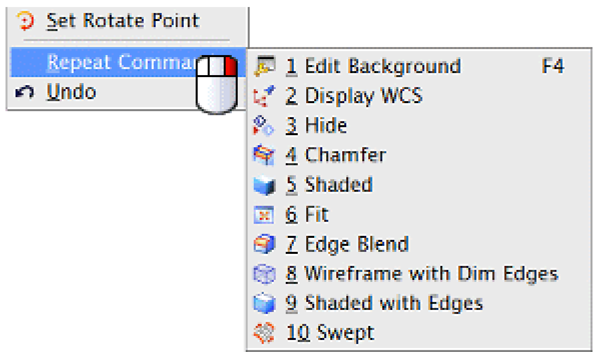
Application	Product Template Studio
Prerequisite	You must add HD3D tags to a product template.

Chapter

3 *Gateway*

Repeat Command

Use **Repeat Command** to rerun any one of the last ten commands that you used.



The last ten commands that you used appear in the **Repeat Command** drop-down list in the order in which they were activated. The most recently activated command appears at the top of the list. When you choose a command from the list, NX validates whether the command can be rerun in the current scenario, and then activates it. For example:

- If you choose the **Extrude** command from the **Repeat Command** list, the **Extrude** dialog box opens.

- If you turned the WCS on the last time you chose the **Display WCS** command, the WCS is turned off when you choose **Display WCS** in the list.

The **Repeat Command** list is cleared when you do any of the following:

- Change applications.

- Close a part.

- Enter or exit a task environment, such as the Sketch task environment.

You can also run the last command that you used at any time when you press F4.

MenuScript users can customize the **Repeat Command** list in the following ways:

- Exclude commands from the list, using the NO_REPEAT keyword in menu files.
- Customize the command name that appears in the list, using the TOOLBAR_LABEL keyword.

Why should I use it?

To reduce the needed number of clicks and mouse movement.

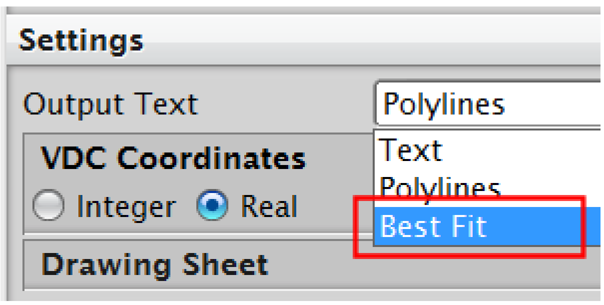
Where do I find it?

Prerequisite	Only commands that appear on the menu bar, toolbars, or shortcut menus that are available in the graphics window, are displayed on the list.
Toolbar	**Repeat Command** **Standard® Repeat Command Drop-down** list
Menu	**Tools® Repeat Command**
Shortcut menu	Right-click in the background of the graphics window® **Repeat Command**

Export CGM enhancement

What is it?

In exported CGM files, you can now output NX fonts as polylines and standard fonts as text using the new **Best Fit** option, in the **Export CGM** dialog box.



Why should I use it?

Using the **Best Fit** option ensures that exported CGM files work well with other CAD applications, which otherwise may not have the capability to render NX fonts.

Where do I find it?

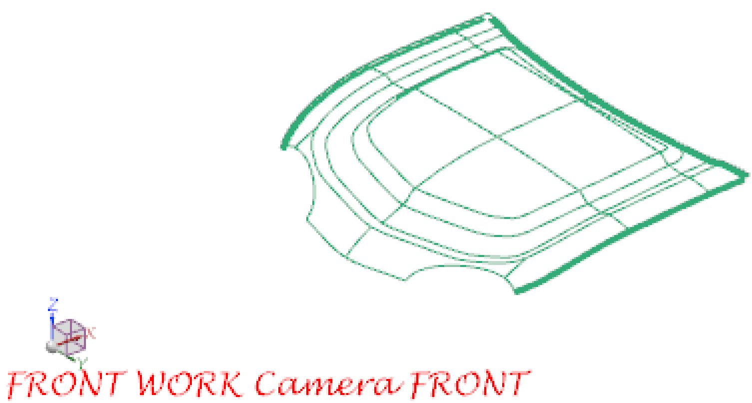
Menu	**File**® **Export**® **CGM**
Location in dialog box	**Export CGM** dialog box® **Settings** group® **Output Text** list® **Best Fit**

Customizing fonts

What is it?

You can now customize the font, the font style, and the font size of the text that appears in the graphics window. For example, you can customize the fonts used for view names, CSYS names, and object names.

You can change the default font to any standard font that is installed on your system. No standard fonts are supplied with NX.



Where do I find it?

Prerequisite	In the **Visualization Preferences** dialog box, on the **Names/Borders** tab, the options to show object names and view names must be selected.
Menu	**Preferences® Visualization**
Location in dialog box	**Visualization Preferences** dialog box® **Color/Font** tab® **Session Settings** group

Shortcuts to common tools options

What is it?

You can now display the list options as shortcuts for the following common tool options in dialog boxes:

- **Specify Point**
- **Specify Vector**
- **Specify Plane**

You can turn on the shortcuts by selecting **Show Shortcuts**. This gives you easy access to the most recently used options.

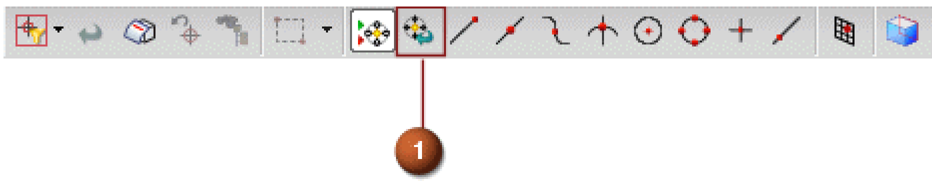
	**Specify Point** example
	**Specify Vector** example
	**Specify Plane** example

Selection enhancements

What is it?

A new option is added to the Selection bar and tooltips are available at the cursor location for objects in the graphics window.

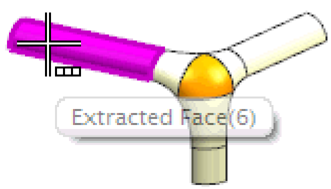
Selection bar option



Clear Snap Point — Turns off all selected Snap Point options on the Selection bar.

Tooltips at the cursor location

You can point to an object to view its name or type in a tooltip that appears at the cursor location. In previous releases, this information was available only on the Status line.



Preselected object with tooltip

Note Tooltips at the cursor location appear translucent in computers running on the Windows XP or Windows 7 operating systems.

The tooltips are visible by default. To turn off these tooltips, clear the **Object Tooltip on Rollover** check box.

Where do I find it?

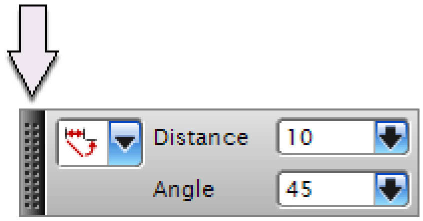
Menu	**Preferences**® **Selection**
Location in dialog box	**Selection Preferences** dialog box® **Highlight** group® **Object Tooltip on Rollover** check box

On-screen input box enhancements

What is it?

Grips

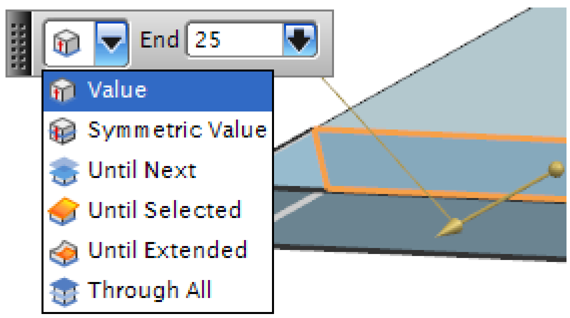
Grips have been added to on-screen input boxes. To move an onscreen-input box, click the grip and drag the box to a new location.



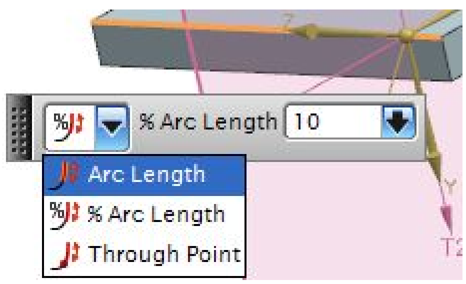
Selection options

Selection options that are common to many dialog boxes are now available from on-screen input boxes.

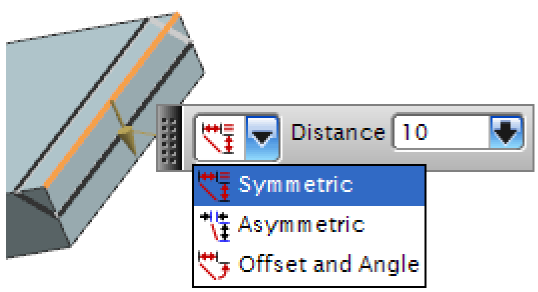
- Limits



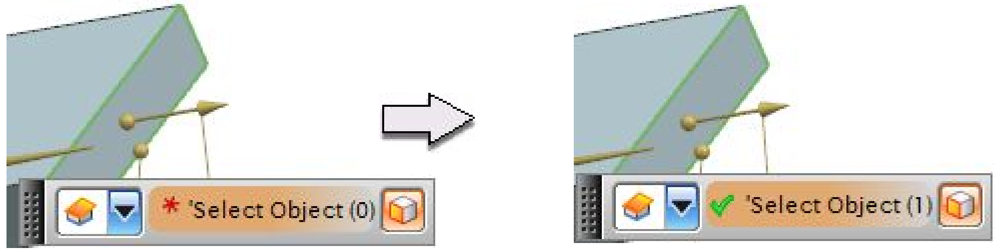
- Location along an arc



- Chamfer types



Display of object selection



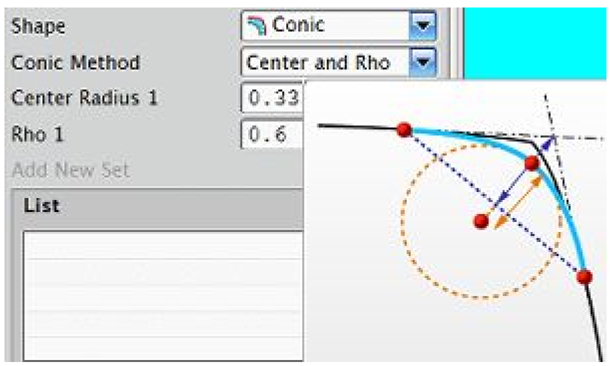
Why should I use it?

Mouse travel from the graphics window to the dialog box is reduced, increasing efficiency.

Balloon tooltips for options

What is it?

Balloon tooltips are now available in some dialog boxes for options that need more information. The tooltip appears when you move your cursor over the label or icon. To turn on these tooltips, select the new **Show Balloon Tooltips on Dialog Options** check box in the **Customize** dialog box.



NX Open users and Block UI Styler users can add balloon tooltips to the dialog box options that they create, using block properties.

Where do I find it?

Menu	**Tools**® **Customize**
Location in dialog box	**Customize** dialog box® **Options** tab® **Tooltips** group® **Show Balloon Tooltips on Dialog Options** check box

Toolbar access for commands

What is it?

You can now easily and quickly access the following commands from their respective toolbars.

Location on toolbar	Name of the command
Active Mockup	**Grid and Work Plane**
Analyze Shape	**Display Non-Proportional Zoom** **Set Non-Proportional Zoom** **Non-Proportional Zoom Options** **Mirror Display** **Set Mirror Plane**
Direct Sketch	**Orient View to Sketch**
Utility® Show/Hide Drop-down list	**Show and Hide**
Utility	**Object Preferences**
View® View Operation Drop-down list	**Fit All Views** **Set View to WCS** **Perspective** **Perspective Options**
View® Rendering Style Drop-down list	**Rendering Style**

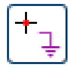
View® View Layout Drop-down list	**Open Layout** **New Layout** **Update Display** **Regenerate All Views** **Replace View** **Delete Layout** **Save Layout** **Save Layout As**
Visualization	**Line Antialiasing** **Facet Settings** **Face Edges** **Facet Edges** **Full Scene Antialiasing** **Translucency** **Random Color Display** **Shuffle Random Color**

Common tool enhancements

What is it?

The Point tool, Vector tool, and Plane tool are now enhanced as follows:

- When you edit an existing point, vector, or plane, NX remembers its type.

  Note A point is displayed as a **Fixed** type  when the point being edited is not associative.

- A new **Constructed** option is available on the Point, Vector, or Plane lists. This option is available when you edit a point, a vector, or a plane that was created by using an option that is not available on the lists. To access all the parameters of a constructed point, vector, or plane, you must use the **Point**, **Vector**, or **Plane** dialog box.

  Example If you use the **Point** dialog box to create an associative point that includes an offset, when you edit that point, the point appears as a **Constructed** point on the Point list.

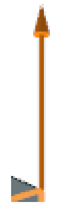
Constructed point icon

Constructed vector icon

Constructed plane icon

- When you choose a type for creating a point, vector, plane, or CSYS, the dialog box remembers your selection when you reopen the dialog box.

- When the tool option in the dialog box is the current selection option, the display of the handle for the tool changes.

Tool Option	Handle display when the tool option is the current selection.	Handle display when the tool option is not the current selection.
Specify Point		
Specify Vector		
Specify Plane	Shape: Plane object Color: Handle selection color Size: Thicker than previous wireframe.	Shape: Plane object Color: Handle color Size: Same as previous wireframe

Attributes enhancements

NX attributes are now enhanced. You can:

- Define part attributes of the null, Boolean, string, integer, number, and date data types.
- Use variable length arrays of supported data types to define values of part attributes.
- Define the unit of measure for a part attribute of the number data type.
- Map attributes between NX and Teamcenter in a better way, as part attributes are no longer restricted to the string data type.
- Edit the set of attributes in component instance objects.
- Access attributes through a single standardized user interface.
- Link expressions to attributes. Any change in the attributes is propagated to the linked expressions.

- Link attributes to expressions. The value of an attribute is calculated by evaluating the linked expression.

  Note Attributes of the date data type cannot be linked to or be linked from expressions.

- Link an attribute in one object to an attribute in another object.
- Use the new **Attribute Templates** command to create and manage attribute templates that you can use to create attributes.

When you use attribute templates, you can:

- Create attribute templates and save them either in a catalog file or in a part file. You can share attribute templates saved in a catalog file across sessions but you can use attribute templates saved in a part file within the part only.
- Standardize attribute identification data, such as title, type, and dimension.
- Store and standardize attribute semantic data, such as a note, or a category.
- Store and standardize data that assists in the value entry of an attribute, such as a default value, a constraint, or a value list.
- Specify whether attributes are to be copied when an object is copied.
- Define an alias for the title of a standard attribute. The alias may use non-English characters.
- Use a mapped attribute to define user access to non-mapped attributes, in Teamcenter Integration.

Where do I find it?

Attribute Templates command

Menu	**File® Utilities® Attribute Templates**

Viewing Files

Rotate enhancements

What is it?

When you use the **Rotate** command, you can now do the following:

- Switch between rotating a model about a set center of rotation and standard rotation by holding Alt and dragging the middle mouse button. In standard rotation, a model is rotated about an arbitrary center of rotation located in the center of the view volume. Previously, you could rotate a model either about a set center of rotation or by using standard rotation, but you could not switch between the two modes of rotation. If you wanted to return to a previously defined center of rotation, you had to redefine it.

- Rotate a model with greater precision when you select the **High Precision Rotation** preference.

High Precision Rotation preference	Provides finer control over rotations in the current session. This preference is a session setting. The default rotation provided when you select this preference is the same as the rotation provided when the environment variable UGII_ROTATION_PRECISION_VALUE = 8. For more precise rotations, specify a value from 1 to 25 for the UGII_ROTATION_PRECISION_VALUE environment variable. A value of 1 provides standard rotation and higher values provide finer rotation.

Note These enhancements apply only to mouse rotation and not to space ball rotation.

Why should I use it?

Fine rotations are extensively used in design evaluation workflows to:

- Study a model and analyze its surface to identify flaws or imperfections.

- Ensure that the product appearance is acceptable from all possible viewing directions.

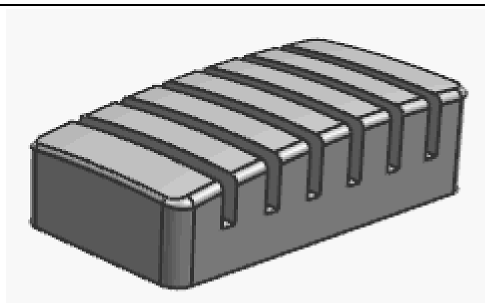
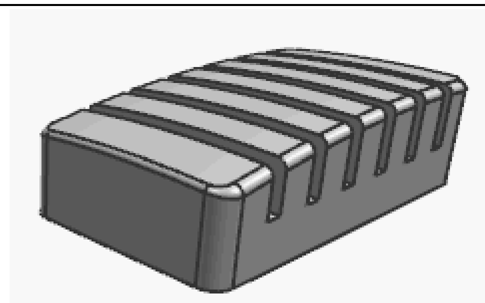
Where do I find it?

Menu	**Preferences® Visualization**
Location in dialog box	**Visualization Preferences** dialog box® **View/Screen** tab® **Session Settings** group® **High Precision Rotation** check box

Perspective Options

What is it?

Use the **Perspective Options** command to control the distance from the camera to the target in a perspective view. The display updates instantly.

 **Trimetric camera far from target**	 **Trimetric camera close to target**

Why should I use it?

Previously, to adjust the perspective you had to use the **Camera® Edit** command, which is slow to start and not as easy to use as the **Perspective Options** command.

Where do I find it?

Prerequisite	You must click **Perspective**  on the **View** toolbar in the **View Operation Drop-down** list or choose **View® Operation® Perspective**.
Toolbar	**View® View Operation Drop-down** list® **Perspective Options** 
Menu	**View® Operation® Perspective Options**

Camera staging view enhancements

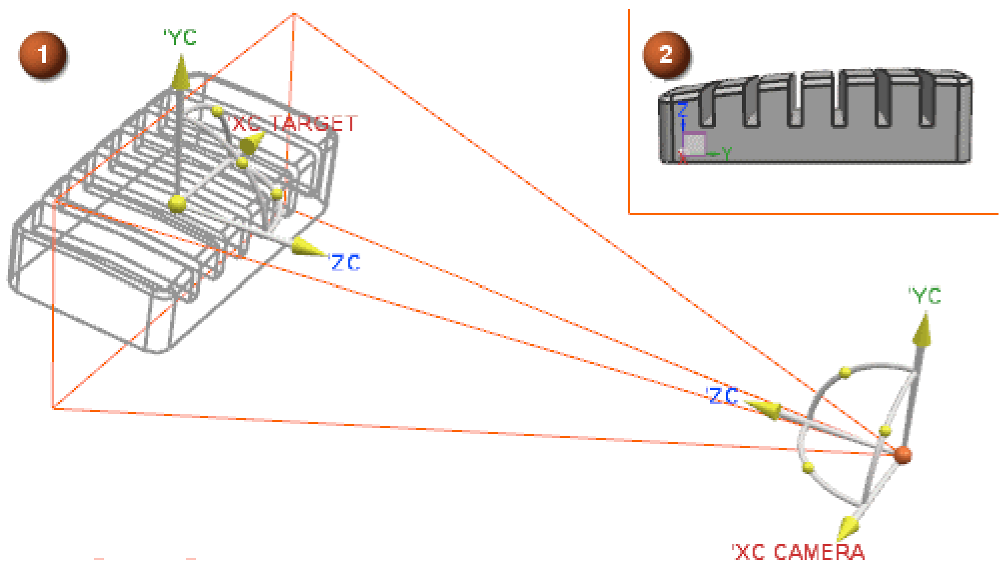
What is it?

The camera staging view has the following enhancements:

- The model is rendered using the **Static Wireframe** rendering style. As objects are displayed using only edge geometry, this reduces the time required to regenerate the model when you change the view.

- The orientation of the model when you enter the camera staging view is now almost the same as the orientation in the original view. Previously, the orientation was not predictable.

- The results view is now opaque and appears on the top right corner of the camera staging view. Therefore, you cannot see any part of the model in the camera staging view which appears in the same space as the results view.



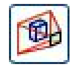
1. Camera staging view with handles for manipulating the target and camera

2. Results view showing the view as seen by the camera

Why should I use it?

You can easily and quickly set up the camera using the enhancements available in the camera staging view.

Where do I find it?

Toolbar	**Visualization® Capture and Edit Camera** 
Menu	**View® Camera**
Part Navigator	**Cameras**

2D planar texture space enhancements

Enhancing 2D planar texture space mapping

What is it?

You can enhance 2D planar texture space mapping by updating the texture space vectors based on the camera direction of the view.

The following options are available on the **Texture Space** tab in the **Material Editor** dialog box:

Camera Direction Plane Automatically adjusts the texture space vectors — normal vector or normal and up vectors — based on the camera direction of the view.

The **Camera Direction Plane** option is similar to the **Arbitrary Plane** option for texture space mapping. When you select either option, texture is mapped onto a user defined plane that is defined with a center point, an up vector, and a vector normal to the plane. The difference is that if you use the **Camera Direction Plane** option, the texture space vectors adjust automatically based on the camera direction of the view. The texture orientation and position update dynamically as you rotate the part. If you use the **Arbitrary Plane** option, you must click **Update Texture to Camera Direction** to rectify the texture space mapping.

Note When you update the texture space based on camera direction, the texture orientation and positioning update dynamically as you rotate the part. This could degrade display performance, especially when the part is large. To delay the update, use the **Disable Dynamic Update Camera Direction Plane Texture Space** option available in the **Visualization Performance Preferences** dialog box. For more information, see Disabling dynamic update for Camera Direction Plane.

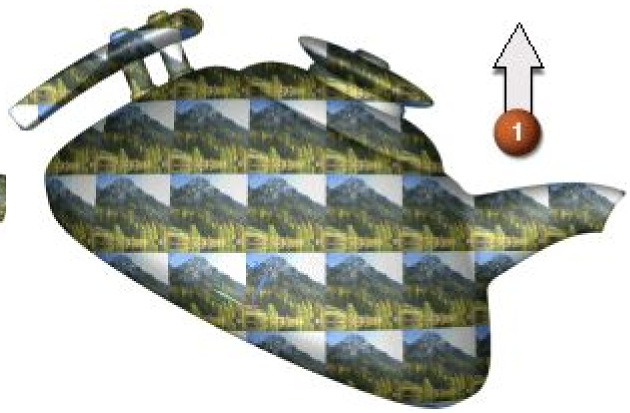
Update Texture to Camera Direction Updates either the texture space normal vector or both normal and up vectors based on the camera direction. Use this option to rectify incorrect texture mapping which appears when you use the **Arbitrary Plane** option.

Arbitrary Plane texture mapping

Arbitrary Plane texture mapping after clicking Update Texture to Camera Direction

Normal Vector and **Normal and Up Vectors** Updates either the texture space normal vector or both normal and up vectors based on the camera direction.

Texture space normal vector only updated based on the camera direction



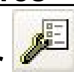
Texture space normal and up vectors updated based on the camera direction

1 = Up Vector

Why should I use it?

Existing texture space types suit regular shaped 3D models. For example, for cylindrical 3D models, such as beer bottles, you can use the **Cylindrical** option for texture space mapping. For irregular shaped 3D models, the **WCS Auto Axis** option provides the best results. However, the **WCS Auto Axis** option produces undesirable results at edges between planes. Use the new options to rectify incorrect texture mapping.

Where do I find it?

Prerequisite	(**Camera Direction Plane**) **Advanced Studio Display** mode must be enabled. (**Normal Vector** and **Normal and Up Vectors**) **Type** must be set to **Arbitrary Plane** or **Camera Direction Plane**. (**Update Texture to Camera Direction**) **Type** must be set to **Arbitrary Plane**.
Toolbar	**Visualize Shape® Materials/Textures** 
Menu	**View® Visualization® Materials/Textures**
Location on dialog bar	**Editor** group® **Launch Material Editor** 
Location in dialog box	**Material Editor® Texture Space** tab

Disabling dynamic update for Camera Direction Plane

What is it?

Use the **Disable Dynamic Update Camera Direction Plane Texture Space** option to delay the update of texture orientation and position based on the camera direction of the view.

When you select the **Camera Direction Plane** option for texture space mapping in the **Material Editor** dialog box, the texture space vectors are automatically adjusted based on the camera direction of the view. The orientation and position of texture updates dynamically as you rotate the part. Use this option to delay the update of texture orientation and position, so that the update occurs only after you release the mouse button. For more information about updating texture space vectors based on camera direction, see Enhancing 2D planar texture space mapping.

Why should I use it?

When you update the texture space based on camera direction, the orientation and positioning of texture updates dynamically as you rotate the part. This could degrade display performance, especially when the part is large.

Where do I find it?

Prerequisite	Rendering style must be set to **Studio**. Advanced Studio Display mode must be enabled. **Camera Direction Plane** must be selected in the **Type** list on the **Texture Space** tab in the **Material Editor** dialog box.
Menu	**Preferences® Visualization Performance**
Location in dialog box	**Visualization Performance Preferences** dialog box® **General Graphics** tab® **Session Settings** group® **Studio Views**

Facet Settings for visualization

Facet Settings for visualization

You can use the **Facet Settings** command to adjust the tolerances used for generating facets for display of exact geometry (solid and sheet bodies) in the graphics window.

You can:

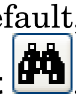
- Modify the current resolution settings to control faceting. You can view, edit, and save the current resolution settings to produce coarser or finer tessellations.

- Maintain a constant view scale to ensure that the facets appear at the same locations on the model, even when you modify the tolerance settings. Previously, the facets appeared at different locations when you zoomed in or out. Now, you can specify a constant facet to view scale to ensure that facets appear at the same location. This is especially useful when you want to evaluate a design.

- Reproduce faceting when you specify a constant view scale. This setting is saved with the part file.

- Prevent your specified tessellation parameters or facet scale from changing by setting **Update** = **None**.

Resolution setting values are predefined in the **Facet Settings** section of the **Customer Defaults** dialog box.

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

To evaluate a design, you might want to make minor adjustments to the default facet settings to isolate a problem on a particular model. You can customize the facet settings to produce tessellations that are either coarser or finer than the default tessellations.

Note Very fine tolerance settings present a smooth surface on points of inflection, but at a high performance cost. As finer tolerance settings could generate millions of facets, NX requires time to produce the tessellations.

The new **Facet Settings** command provides new facet setting options for visualization and easy access to the following options:

- Facet settings that were previously available only in the **Part Settings** group on the **Faceting** tab in the **Visualization Preferences** dialog box.

- Facet setting customization options that were previously available only in the **Customize Tolerance Settings** dialog box.

Where do I find it?

Toolbar	**Visualization® Facet Settings** 
Menu	**View® Operation® Facet Settings**

Note All options available in the **Facet Settings** dialog box are also available in the **Visualization Preferences** dialog box® **Faceting** tab® **Part Settings** group.

Facet Settings options for visualization

What is it?

The new **Facet Settings** command provides new facet setting options and easy access to the following options:

- Facet settings that were previously available only in the **Part Settings** group on the **Faceting** tab in the **Visualization Preferences** dialog box.

- Facet setting customization options that were previously available only in the **Customize Tolerance Settings** dialog box.

The following options are either renamed or are newly available in the **Facet Settings** dialog box.

Shaded Views and **Advanced Visualization Views** group		
New name	Old name	Description
Resolution	**Tolerance** This has a new option called **User defined**.	
Update	**Update Mode**	
Facet to View Ratio	This is a new option.	Adjusts the facet to view scale. **Automatic**: Automatically adjusts faceting based on the view scale. When you zoom in or zoom out, the location of facets on the model changes. This is the default value. **User Defined**: Maintains a constant facet to view scale specified by **Facet Ratio**.
Facet Ratio	This is a new option.	Available when **Facet to View Ratio** is set to **User Defined**. Specifies a constant facet to view scale to ensure that facets appear at the same locations on the model even when you zoom in or zoom out. Specify a ratio up to ten times the actual basic setting. This setting is saved in the part file, so that you can reproduce the same faceting when required.
Resolution Tolerances subgroups		
New name	Old name	Description
Edge	**Edge Tolerance**	
Face	**Face Tolerance**	
Angle	**Angle Tolerance**	
Width	**Width Tolerance**	

Reset	This is a new option.	Resets the resolution tolerance values of the selected **Resolution** setting to the values specified in the customer defaults.

Note If you select the **Render Solids Using Stored Facets** check box in the **General Graphics** tab of the **Visualization Performance Preferences** dialog box the shaded views are rendered using stored facets. This overrides the settings in the **Shaded Views** group in the **Facet Settings** dialog box and the specified values of tolerance are ignored.

Use one of the following methods to update the stored facets:

- Select **View**® **Layout**® **Update Display**.
- In the **Visualization Performance Preferences** dialog box, click the **General Graphics** tab and in the **Shaded Views** group clear the **Render Solids Using Stored Facets** check box.

Why should I use it?

You can use the **Facet Settings** command to easily control faceting and modify the attributes of the current resolution.

Where do I find it?

Facet Settings command

Toolbar	**Visualization**® **Facet Settings** 
Menu	**View**® **Operation**® **Facet Settings**

Part Settings group

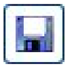
Menu	**Preferences**® **Visualization**
Location in dialog box	**Visualization Preferences** dialog box® **Faceting** tab® **Part Settings** group

Render Solids Using Stored Facets check box

Menu	**Preferences**® **Visualization Performance**
Location in dialog box	**Visualization Performance Preferences** dialog box® **General Graphics** tab® **Shaded Views** group

Update Display command

Menu	**View**® **Layout**® **Update Display**



Customizing a command dialog box

Use the **Save Favorite** and **Save Favorite As** commands to save a single or multiple versions of a command dialog box. If there are specific values and layouts of a dialog box that you use frequently, for instance if you expand or collapse certain groups, then you can set up the dialog box as you frequently use it, and save it as a favorite. This saves you from re-entering values for often-used cases.

Note You cannot save on-screen selections that you make in NX as favorites.

You cannot save dialog favorites in Teamcenter.

You can:

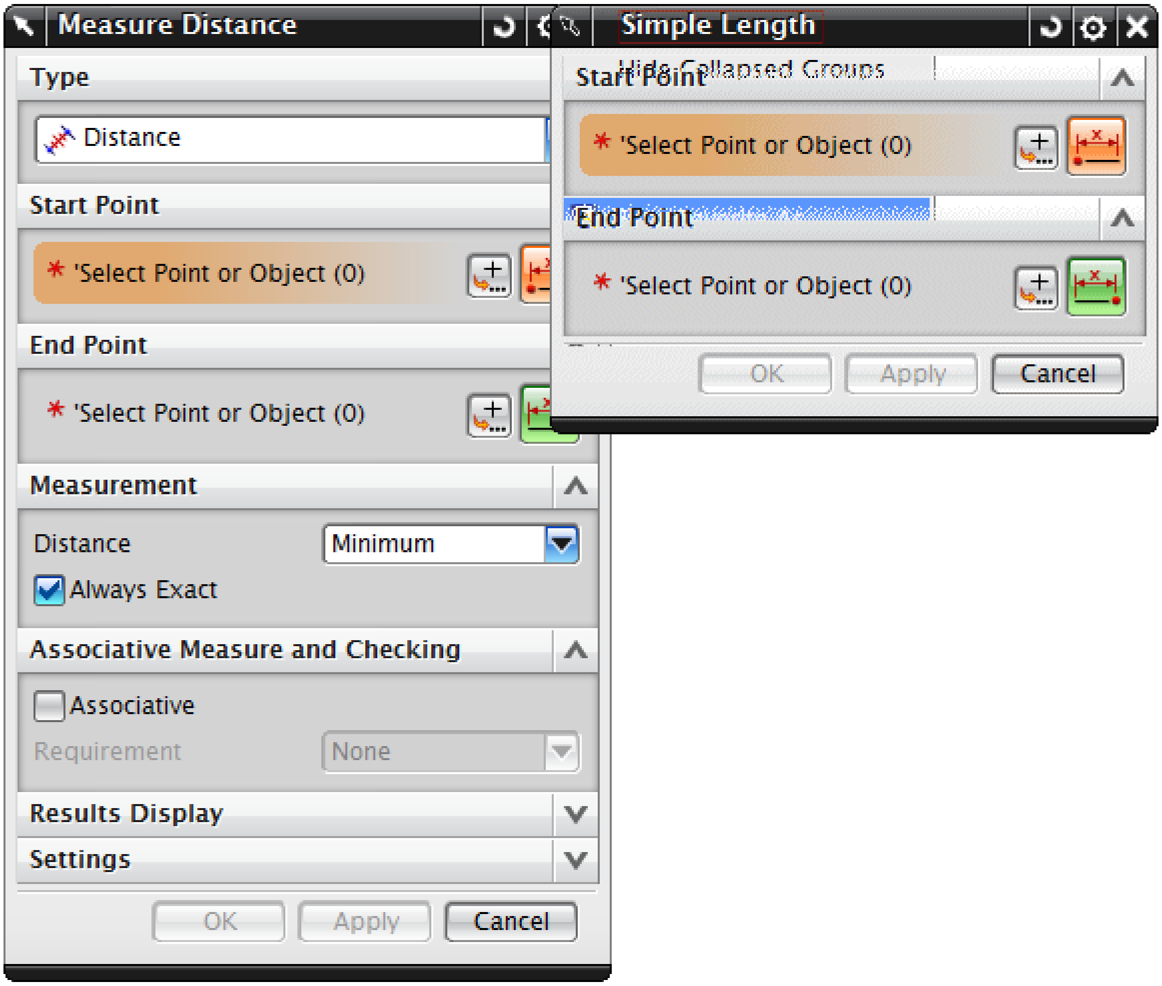
- Save multiple versions of the same command.

- Use the saved favorite in current and future NX sessions.

- Access the saved favorite from the role that you created it in. The saved favorite is added to the role that you save it in.

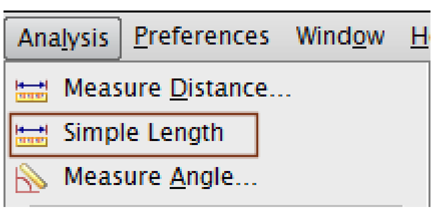
- Open the saved favorite in existing commands.

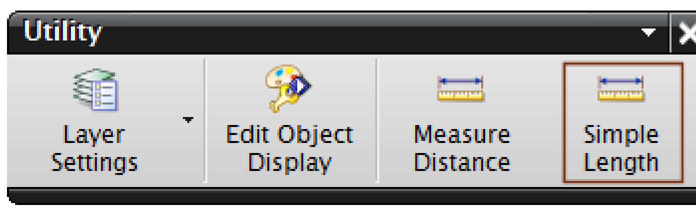
  For example, you can customize the **Measure Distance** command to create a new layout by displaying only the start and end points and hiding all the other groups in the **Measure Distance** dialog box.



- Add the saved favorite to a toolbar or a menu for easy and direct access.

  For example, you can load the customized version of the **Measure Distance** command in the **Analysis** menu or on the **Utility** toolbar.





Where do I find it?

Location in dialog box	Click **Show Menu** on the dialog title bar® **Save Favorite / Save As Favorite**

True Studio Task

True Studio Task

What is it?

You can use the **True Studio Task** environment together with the **Advanced Studio Display** mode for easy setup and realistic visualization of your model and to generate static images.

Note **Advanced Studio Display** mode requires a Studio Visualize license.

You can:

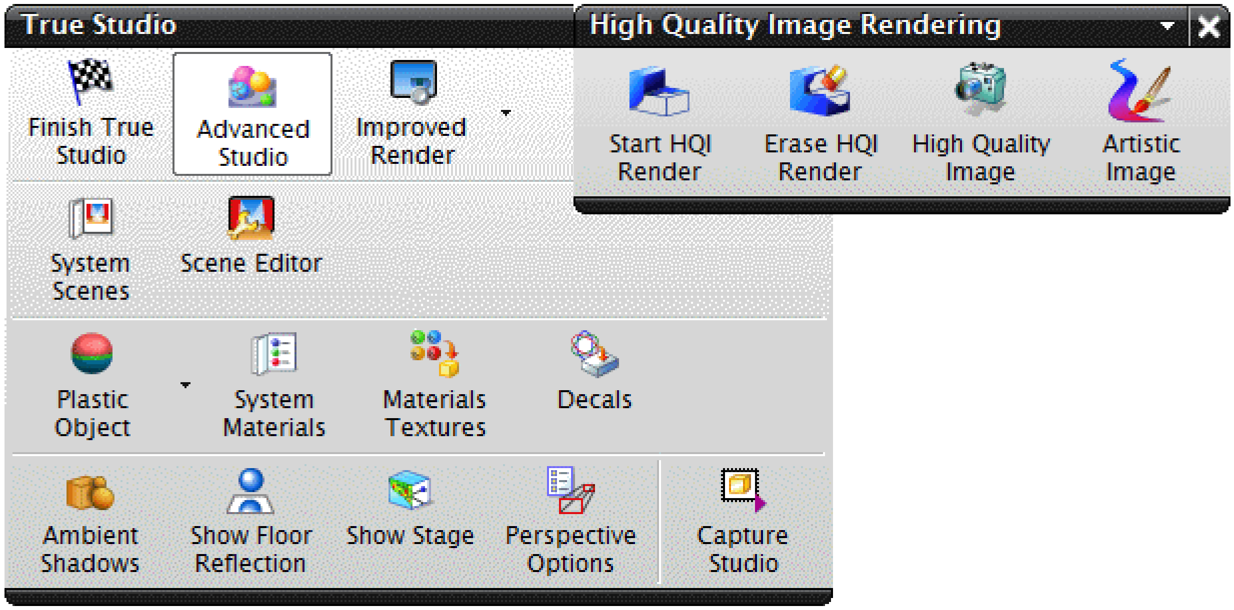
- Generate **High Quality Images**.

  Note **High Quality Images** require a Studio Render license.

- Access all **Advanced Studio Display** mode features.
- Have access to all visualization tools in NX in one place.
- Generate static images of real-time display using **Capture Studio Image**.

You can also add effects of:

- **System Scenes**
- **Advanced Studio Ambient Shadows**
- Different **System Materials** and Studio Global Materials
- Advanced Studio **Image-Based lighting**
- Advanced Studio Render Methods



Where do I find it?

Toolbar	**Visualize Shape** toolbar
Menu	**View® Visualization® True Studio Task**

Studio Global Materials

Use Studio Global Materials to simplify material set up in **Studio**, **Advanced Studio** and **High Quality Image** rendering modes. You can control the Studio Global Materials in the **True Studio Task** environment.

NX displays a Studio Global Material on the model geometry when no material is assigned.

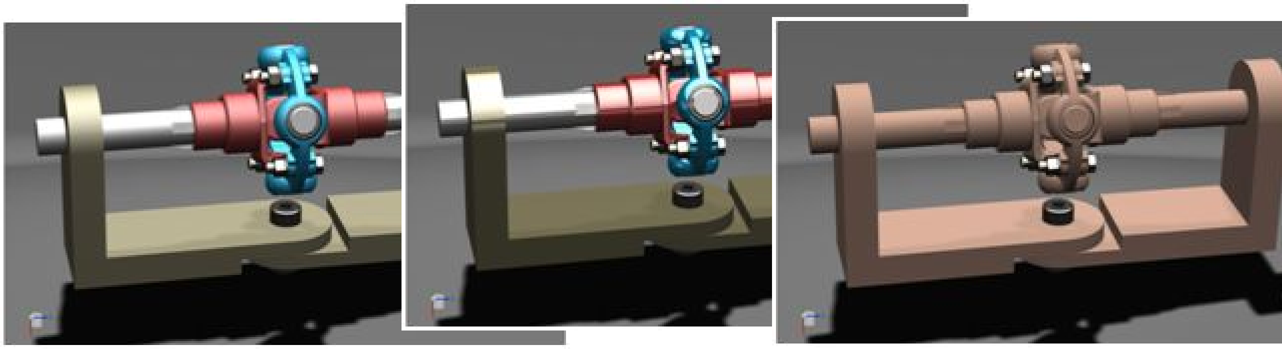
You can choose from the following global materials:

- **Plastic Object Colors**
- **Shiny Metal Colors**
- **Clay**
- **Plasticine**

- **Chrome**

You can customize the default Studio Global Material by using the **Studio Global Material** customer default.

Colorwash Global Materials use the object color along with the material characteristics, such as shiny metal, when displaying the model geometry.



Plastic colorwash **Shiny metal colorwash** **Clay**

Where do I find it?

Studio Global Material

Prerequisite	**True Studio Task** needs to be selected from the **Visualize Shape** toolbar.
Toolbar	**True Studio® Studio Global Material Drop-down** list

Studio Global Material customer defaults:

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Gateway® Visualization® Materials/Texture** tab

Advanced Studio Ambient Shadows

Use the **Advanced Studio Ambient Shadows** command to produce more realistic shadows for advanced studio views. This command produces shadows by considering the ambient lighting of the entire scene, instead of a single light source. It adds depth to the model and helps you perceive the 3D shape of the model better. Areas that are partially hidden by other objects, such as convex corners, are darkened.

Shadows rendered considering the ambient lighting

Shadows rendered using a single light source

When you use the command, you can:

- Create new visualization scenes with different Advanced Studio Ambient Shadows settings.

- Edit the **Advanced Studio Ambient Shadows** settings in the **Scene Editor** dialog box to produce optimal results for the part. The results are displayed dynamically. The settings are saved with the part.

- Delay the update of the shadows by using the **Disable Dynamic Update Advanced Studio Ambient Shadows** Visualization preference.

  Note The real time ambient shadow rendering could degrade display performance, especially when the part is large and if you are not using high end graphics devices. You can use this preference to delay the update of shadow orientation and position, so that the update occurs only at the end of dynamic viewing.

Where do I find it?

Prerequisite	**Advanced Studio Ambient Shadows** is supported only on devices that support the **Advanced Studio Display** mode and requires availability of DirectX 10. The **Advanced Studio Display** mode must be enabled. On the **Visualize Shape** toolbar, click **Advanced Studio Display** , or choose **View® Visualization® Advanced Studio Display** to enable the **Advanced Studio Display** mode. **Note** If your device does not support Ambient Shadows, you will get an error message stating the same.

Advanced Studio Ambient Shadows command

Toolbar	**Visualize Shape® Advanced Studio Ambient Shadows** 

Advanced Studio Image-Based Lighting

What is it?

Use Advanced Studio Image-Based Lighting to add realistic lighting to 3D scenes.

Image-Based Lighting uses light from the surrounding scene during rendering to create more realistic display.

Note This functionality is available only on devices that have the **Advanced Studio Display**  mode available and enabled.

Image-based Lighting: Off
Notice the harsher lighting from the default scene lights directed from the left and right

Image-based Lighting: On
Notice the softer lighting from the light boxes in the environment image

Why should I use it?

Use this to make **Advanced Studio Display** rendering more realistic and to match HQI rendering better.

Where do I find it?

Prerequisite	The **Advanced Studio Display** mode must be enabled.
Menu	**View® Visualization® Scene Editor**
Toolbar	**Visualize Shape® Scene Editor**
Location in dialog box	**Global Illumination** tab

Car paint material shader

What is it?

Use this material shader to simulate the effects of paint used in the automobile industry.

The Car Paint material shader has the following components:

- Base color
- Metallic flake
- Lacquer

You can use the car paint material shader only in:

- **Advanced Studio** real-time display mode.
- **High Quality Image** static rendering.

A new set of **System Materials** based on the Car Paint material shader:

Why should I use it?

Use car paint material shaders for realistic rendering of parts.

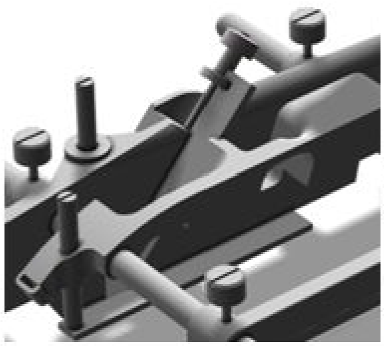
Where do I find it?

Toolbar	**True Studio® Materials/Textures** ® **Material Editor** dialog box
Menu	**View® Visualization® Materials/Textures® Material Editor** dialog box
Location in dialog box	**General® Type® Car Paint**
Resource bar	**System Materials** ® **automotive** folder

Advanced Studio Render Methods

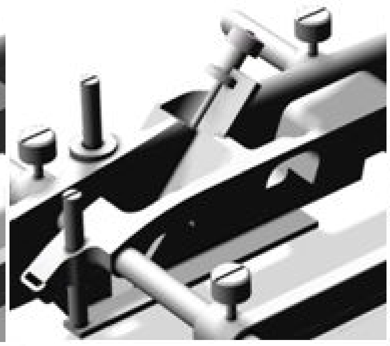
What is it?

Use the **Advanced Studio** Render Methods to choose quality and performance trade-offs for the **Advanced Studio Display** mode.



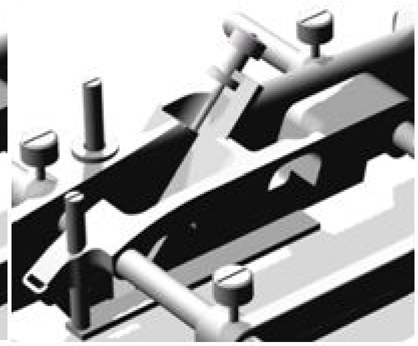
Full Render

Provides the best quality display by including all Advanced Studio effects.



Improved Render

Provides a fast display by eliminating some Advanced Studio effects.



Preview Render

Provides a fast quality display by eliminating many Advanced Studio effects.

The following features are present in the **Advanced Studio Display** mode, depending on the Render Method selected:

Advanced Studio Rendering Feature	Full Render	Improved Render	Preview Render
Soft Shadows	✓	✓	
Soft Shadows during dynamics	✓	✓	
Ambient Shadows	✓	✓	
Ambient Shadows during dynamics	✓	✓	
Image-Based Lighting	✓		
Image-Based Lighting during dynamics	✓		
Floor Reflections	✓	✓	✓
Floor Reflections during dynamics	✓	✓	
Entire update during dynamics	✓		

Note

- Default Render Method is set to **Improved Render**.
- Individual UI controls do not change depending on the Render Method. You will be warned when a low Render Method is enabled.

- **Visualization Performance Preference** settings take precedence over **Advanced Studio** render settings.

  For instance, the Image-Based Lighting dynamic update is supported in **Full Render** mode. However if **Disable Dynamic Update Advanced Studio Image-Based Lighting** is selected in the **General Graphics** tab in **Visualization Performance Preference**, then you cannot get dynamic update of Image-Based Lighting.

Where do I find it?

Prerequisite	**Advanced Studio Display** must be enabled. **True Studio Task** must be enabled.
Toolbar	**True Studio**® **Advanced Studio Render method** drop-down list® **Full Render** **True Studio**® **Advanced Studio Render method** drop-down list® **Improved Render** **True Studio**® **Advanced Studio Render method** drop-down list® **Preview Render**

System Materials

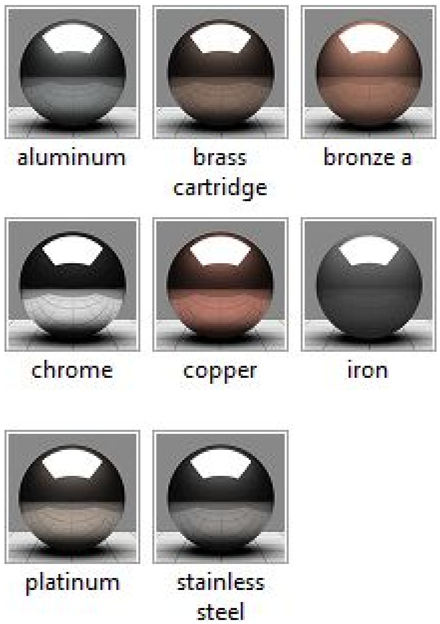
What is it?

You can use a new set of **System Materials** provided in NX. Use these materials to work together with the set of **System Scenes** for realistic visualization.

You can:

- Apply or drag a material from this palette to your selection. This automatically copies the material to the **Materials in Part** palette.

- Copy materials from this palette to the **Materials in Part** palette and edit to make your own material.



Metal

Plastic

Where do I find it?

Prerequisite	A part must be loaded or **True Studio Task** needs to be selected from the **Visualize Shape** toolbar.
Toolbar	**True Studio® System Materials**
Resource bar	**System Materials** tab

System Scenes

You can use a new set of **System Scenes** provided in NX. Use these scenes to work together with the set of **System Materials** for realistic visualization.

You can:

- Use the **System Scenes** for high-end rendering techniques such as global illumination with image-based lighting.
- Use the **Scene Editor** dialog box to make further changes to your background, scenes, lighting, and reflection.

- Applying system visualization scenes to your model overrides all existing scene characteristics such as lights, shadows, and environment.

- Use the **True Studio Task** Render Methods to define quality and performance trafe-offs.

Grey Studio 1

Granite Counter

Where do I find it?

Prerequisite	A part must be loaded or **True Studio Task** needs to be selected from the **Visualize Shape** toolbar.
Toolbar	**True Studio® System Scenes**
Menu	**Studio® System Scenes**
Resource bar	**System Scenes**

Chapter

4 CAD (Design)

Sketching

Direct Sketching enhancement

The **Direct Sketch** toolbar is now available in the Shape Studio, and Sheet Metal applications.

The following Sketch commands are now available on the **Direct Sketch** toolbar:

-  **Project Curve**
-  **Intersection Curve**
-  **Intersection Point**
-  **Trim Recipe Curve**

Why should I use it?

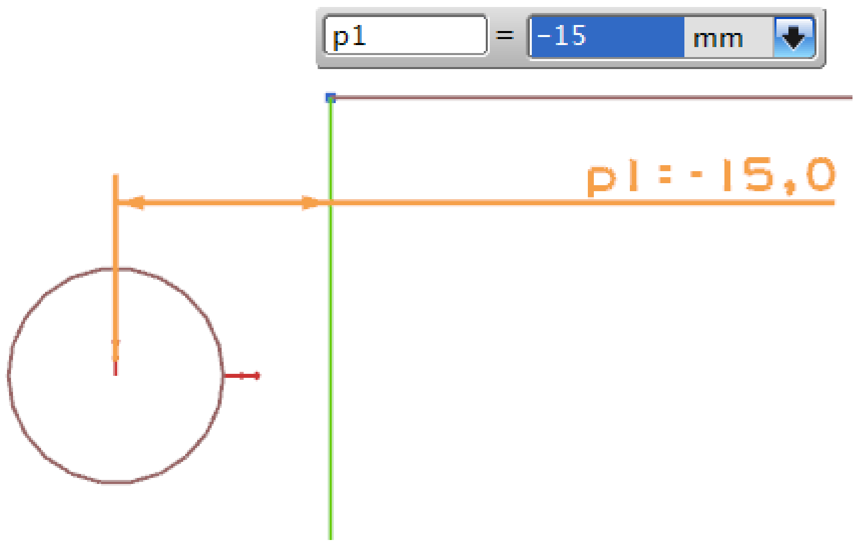
Use the commands on the **Direct Sketch** toolbar to create a sketch on a plane without entering the Sketch task environment. Direct sketching requires fewer mouse clicks, which makes creating and editing sketches faster and easier.

Where do I find it?

Application	Modeling, Shape Studio, Sheet Metal
Toolbar	**Direct Sketch**

Sketch dimension enhancement

Perpendicular, Horizontal, Vertical, and Angular dimensions maintain their direction when the expression value is set to zero. You can also enter negative values for these dimension types to achieve the same results as using the **Alternate Solution** command.



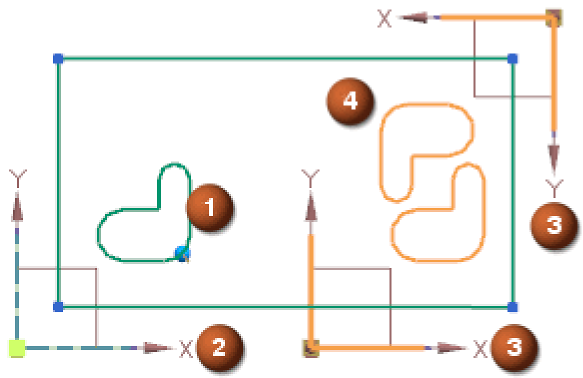
Where do I find it?

Application	Modeling, Drafting, Shape Studio, Sheet Metal
Toolbar	(Modeling, Shape Studio, Sheet Metal) **Direct Sketch** (Sketch task environment and Drafting) **Sketch Tools**
Menu	(Modeling, Shape Studio, Sheet Metal) **Insert® Sketch Constraint® Dimensions** (Sketch task environment) **Insert® Dimensions** (Drafting) **Insert® Dimensions**

Sketch General Pattern

You can use the new **General Pattern** option in the **Pattern Curve** dialog box to pattern a set of sketch curves:

- From a point to one or many points
- From a CSYS to one or many CSYS



1 Selected curve for pattern.

2 Selected **From Coordinate System**
From a point or CSYS.

3 Selected **To Coordinate Systems**
To a point or coordinate system or many points or coordinate systems.

4 **Orientation** = **Follow Pattern**

You can use the **Lock Orientation** option in the **Pattern Curve** dialog box to lock the rotational angle constraint to follow the original curves. If you do not select the option, you can change the rotation angle of the entire pattern.

Where do I find it?

Application	Modeling, Drafting, Shape Studio, Sheet Metal
Toolbar	(Modeling, Shape Studio, Sheet Metal) **Direct Sketch® Pattern Curve** (Sketch task environment and Drafting) **Sketch Tools® Pattern Curve**
Menu	(Modeling, Shape Studio, Sheet Metal, Drafting) **Insert® Sketch Curve® Pattern Curve** (Sketch task environment) **Insert® Curve from Curves® Pattern Curve**
Location in dialog box	**Pattern Definition** group® **Layout** list® **General**

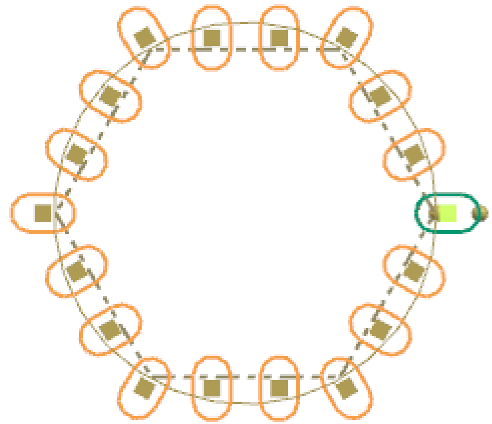
Sketch Pattern enhancement

There are four new pattern options in the **Pattern Curve** dialog box to pattern a set of sketch curves. These options are especially useful when you add sketch curves to a drawing.

These additional pattern curve options are only available if you disable **Create Inferred Constraints** .

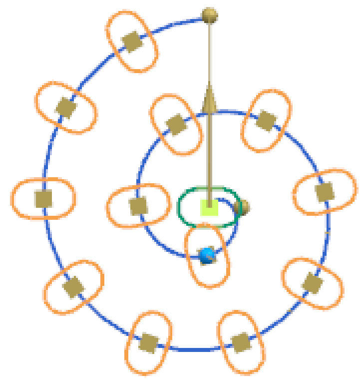
Polygon

Creates an equilateral polygon, defined by an origin, the number of sides, and a radial component defined by the object to pattern on the corner.



Spiral

Creates an **Along** pattern where simplified control of the spiral is provided. The object to pattern will always be at the origin of the spiral. You can specify the distance between the coils, left or right handedness, the total number of turns, and a rotation angle.

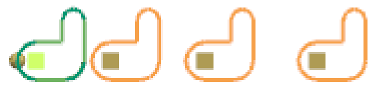


Reference

Creates a pattern based on another pattern.

Pattern Increment

Increases or decreases the distance between each pattern instance by the value specified. **Pattern Increment** can be used with the **Linear**, **Circular**, **Polygon**, **Spiral**, and **Along** layouts.



Where do I find it?

Application	Modeling, Drafting, Shape Studio, Sheet Metal
Prerequisite	Disable **Create Inferred Constraints**
Toolbar	(Modeling, Shape Studio, Sheet Metal) **Direct Sketch® Pattern Curve** (Sketch task environment and Drafting) **Sketch Tools® Pattern Curve**
Menu	(Modeling, Shape Studio, Sheet Metal, Drafting) **Insert® Sketch Curve® Pattern Curve** (Sketch task environment) **Insert® Curve from Curves® Pattern Curve**
Location in dialog box	**Pattern Definition** group® **Layout** list

Delaying Sketch updates

Use the **Delay Update During Edit of Sketch** to delay updating the model from the sketch while sketching directly in Modeling, Shape Studio, or Sheet Metal.

You can then use the **Update Model from Sketch** command to update the model to reflect all the changes you made to the sketch.

Why should I use it?

When you edit a sketch directly in an application, the dependent features are updated for every sketch modification you make. You can use these commands to streamline the editing of a sketch that has many dependant features.

You can first implement a series of sketch edits and then see the results of the edits. This creates a streamlined workflow because you update the model when you are ready.

Where do I find it?

Application	Modeling, Shape Studio, Sheet Metal
Toolbar	**Direct Sketch® Delay Update During Edit of Sketch** **Direct Sketch® Update Model from Sketch**

Menu	**Tools® Update® Delay Update During Edit of Sketch** **Tools® Update® Update Model from Sketch**

Group commands

What is it?

The terminology and interaction of **Group** commands is now consistent across all NX applications.

- You can now create sketch groups when working in Drafting.
- When a sketch is in Modeling and in Drafting, you can select curves and dimensions and group them using the shortcut menu.
- In the **Part Navigator**, in the Drafting application, sketch groups now appear under the sketch node in the sheet and the view sketch leaves of the tree hierarchy.
- Group labels and their function are now as follows in all applications:
  - o **New Group** — Creates a new regular group
  - o **New Sketch Group** — Creates a sketch group
  - o **New Active Sketch Group** — Creates a new empty active sketch group

Where do I find it?

Application	Modeling, Shape Studio, Drafting, Sheet Metal
Menu	**Format® Group® New Group** **Format® Group® New Active Sketch Group**

Synchronous Modeling

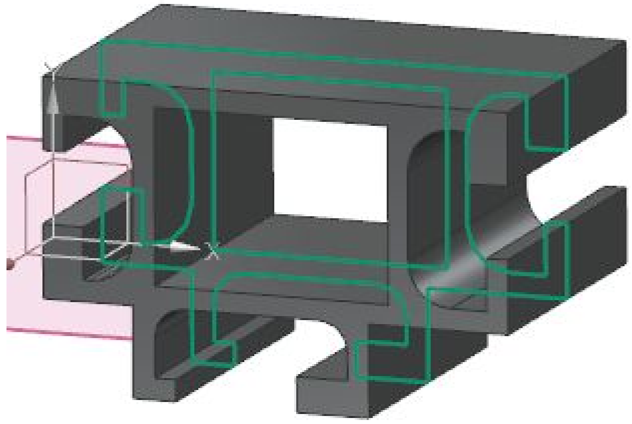
Cross Section Edit enhancement

You can now use the **Edit Cross Section** command in **History Mode**. The sketch of the cross section is saved and you can use this for further edits.

When you section a model, you can:

- Section multiple bodies.
- Select individual faces.
- Exclude holes and blends.

- Add sketch constraints.

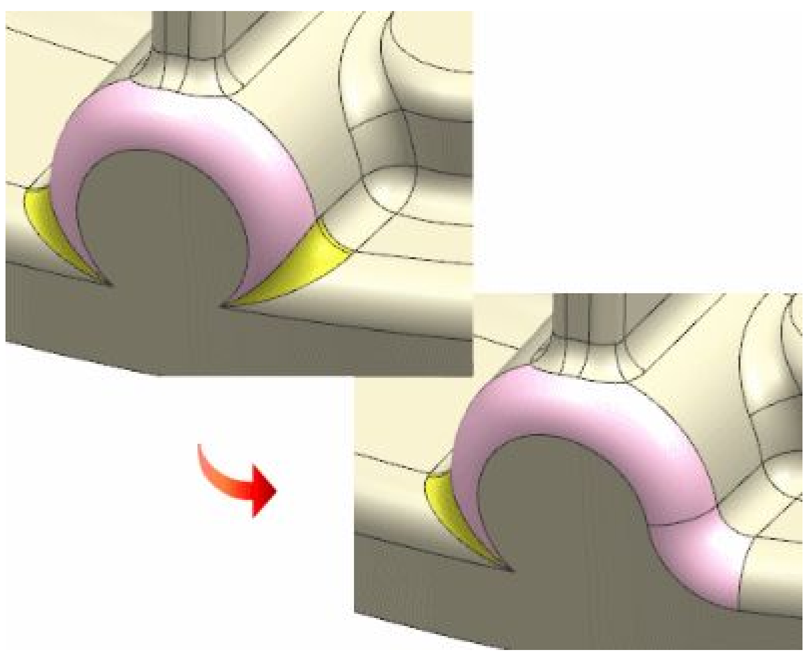


Where do I find it?

Application	Modeling, Shape Studio, Manufacturing
Toolbar	**Synchronous Modeling® Edit Cross Section** 
Menu	**Insert® Synchronous Modeling® Edit Cross Section**

Reorder Blends

Use the **Reorder Blends** command to change the order of two intersecting blends of opposite convexity. For example, from “B overflows A” to “A overflows B”.



Why should I use it?

You can reorder blends:

- When re-creating the blends in a different order would be too time consuming.
- To change the overflow of two blended edges in just one area.
- In parts with or without a feature history.

Where do I find it?

Application	Modeling, Shape Studio, Advanced Simulation, and Manufacturing
Toolbar	**Synchronous Modeling® Reorder Blends** 
Menu	**Insert® Synchronous Modeling® Detail Feature® Reorder Blends**

Delete Face enhancement

When you use the **Delete Face** command you can now delete a face without healing neighboring faces when you clear the new **Heal** check box. If you delete a face on a sheet body, the result is a sheet body with open edges. If you delete a face on a solid body, the solid body is converted to a sheet body.

Without the **Heal** option With the **Heal** option

Where do I find it?

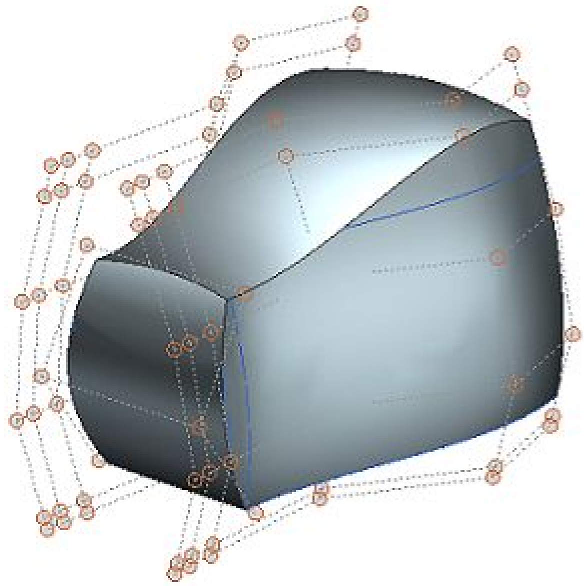
Application	Modeling, Shape Studio, Advanced Simulation, Manufacturing
Toolbar	**Synchronous Modeling® Delete Face** 
Menu	**Insert® Synchronous Modeling® Delete Face**
Location in dialog box	**Settings** group® **Heal** check box

Improved surfaces when extending freeform faces

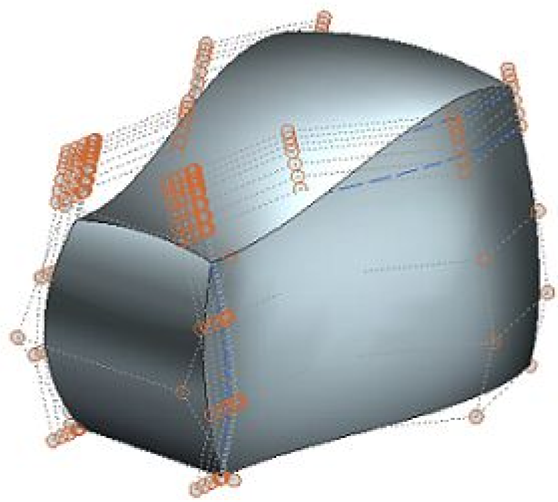
What is it?

Making face modifications using some **Synchronous Modeling** commands, such as **Move Face**, **Make Coplanar**, and **Linear Dimension**, can now result in extended surfaces that are of substantially improved quality.

Setting the new **Surface Extension Method** preference to **Soft** extends faces smoothly as curvature continuous C2 surfaces.



Setting the **Surface Extension Method** preference to **Linear** extends faces using the original NX algorithm, which produces a less smooth surface with additional, unnecessary C0 knots and surface poles.



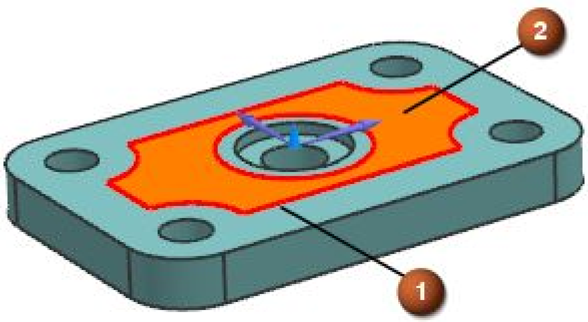
Where do I find it?

Application	Modeling
Menu	**Preferences**® **Modeling**
Location in dialog box	**Freeform** tab, **Surface Extension Method** set to **Soft**

Face Region Boundary enhancement

What is it?

You can now use the **Face Rule** option **Region Boundary Faces** for selecting faces in **History** mode when using the **Pull Face** and **Offset Region** commands.



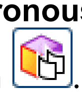
1 Curves on top face of part

2 Portion of face selected using **Region Boundary Faces**

Why should I use it?

When you are in **History** mode and need to **Pull** or **Offset a Region** of faces, and need to select a portion of a face limited by curves.

Where do I find it?

Application	**Modeling, Shape Studio**
Selection bar	**Face Rule® Region Boundary Faces**.
Prerequisite	**Synchronous Modeling→Pull Face**  or **Offset Region** .

Interpart selection in Synchronous Modeling

What is it?

You can now use the following Synchronous Modeling commands to select interpart objects:

- **Make Coplanar**
- **Make Coaxial**
- **Make Symmetric**
- **Make Tangent**
- **Make Parallel**

- **Make Perpendicular**
- **Paste Face**
- **Linear Dimension**
- **Angular Dimension**
- **Replace Face**

You can specify interpart selections using the **Selection Scope** and **Create Interpart Link** options on the Selection bar.

Where do I find it?

Application	Modeling
Toolbar	Selection bar

Move Face direction and location enhancements

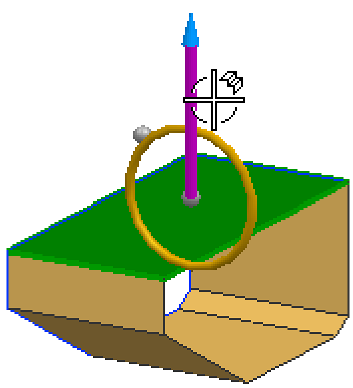
What is it?

You can now set direction and location parameters directly from the graphics window when working with the **Move Face** command.

After selecting a face to move you can:

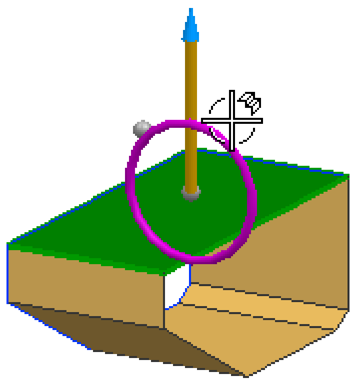
Set the linear direction

Click the distance axis (stem) of the dimension handle and then define a direction by clicking an object or using **OrientXpress**.



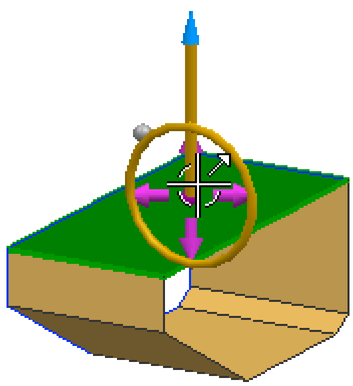
Set the angular direction

Click the angle axis (hoop) of the dimension handle and then define a direction by clicking an object or using **OrientXpress**.



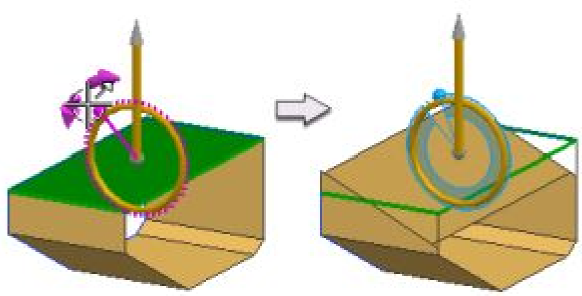
Set the handle location

Move the combined linear/angle handle to a new location by clicking its point and either selecting or dragging to a new point.

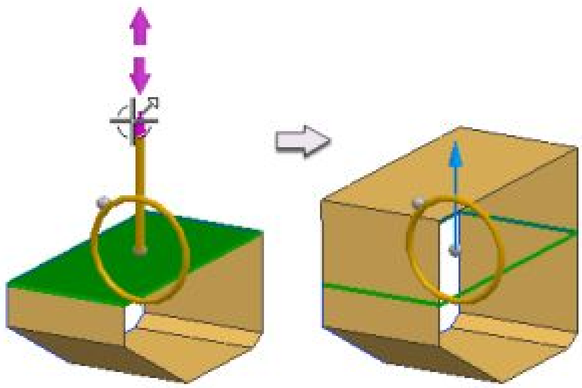


Angular and linear transforms

Drag the angle handle on the hoop to transform the selected face in the angular direction.



Drag the distance handle to transform the face in the linear direction.



Note The angle axis hoop appears when the linear direction is parallel with a global axis (x, y, or z).

Why should I use it?

You can now interact directly with the objects in the graphics window without using the dialog box to set the options.

Where do I find it?

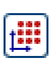
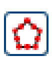
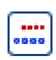
Application	Modeling
Toolbar	**Synchronous Modeling® Move Face** 
Menu	**Insert® Synchronous Modeling® Move Face**

Modeling

Pattern Feature

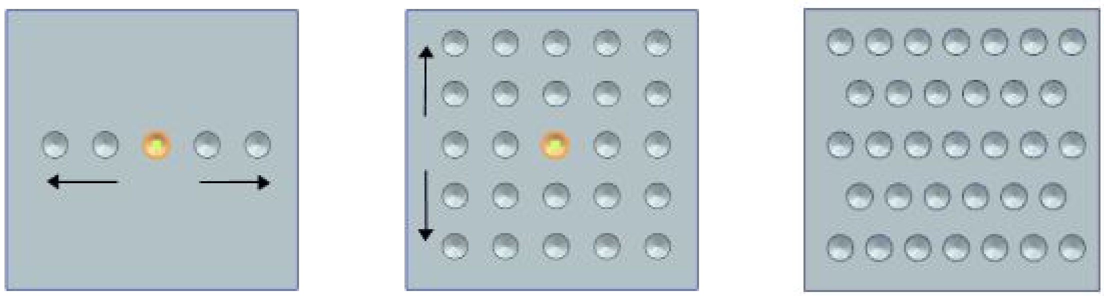
Use the **Pattern Feature** command to create patterns of features (linear, circular, polygon, etc.) with various options for defining pattern boundaries, orientation of instances, clocking and variance.

- You can create pattern features using a variety of pattern layouts.

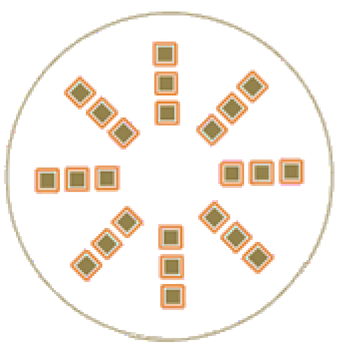
 Linear	 Polygon	 Along	 Reference
 Circular	 Spiral	 General	

- You can fill a specified boundary with a pattern feature.

- For a **Linear** layout, you can specify a **Symmetric** pattern in one or both directions. You can also specify to **Stagger** columns or rows.

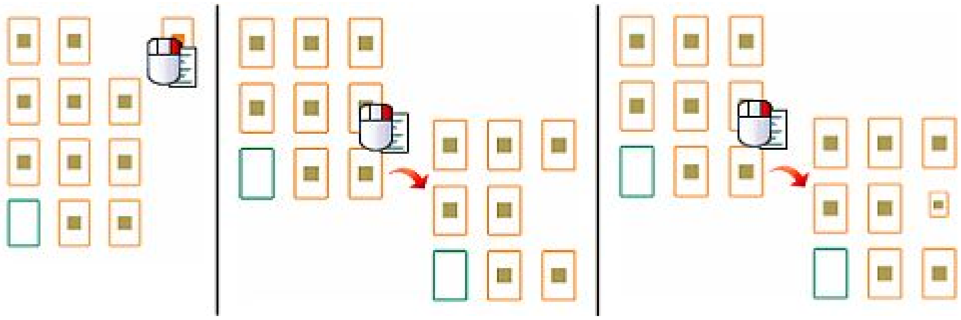


- For a **Circular** or **Polygon** layout, you can choose to radiate a pattern.

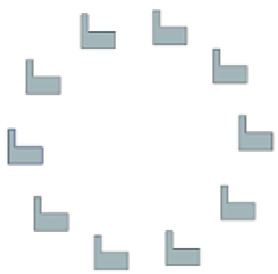


- You can define a **Pattern Increment** by using expressions to specify pattern parameters.
- You can export pattern parameter values to a spreadsheet and make positional edits that are propagated back to your pattern definition.

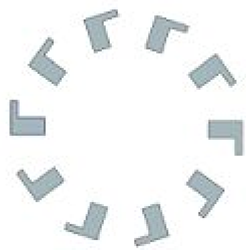
- You can explicitly select individual instance points for clocking, suppression and variance of pattern features.



- You can control the orientation of a pattern.



Orientation same as input

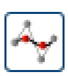


Orientation follows pattern (circular)

- You can choose between **Simple** and **Variational** pattern methods.

Where do I find it?

Application	Modeling and Shape Studio
Prerequisite	History mode
Toolbar	**Feature® Pattern Feature** 
Menu	**Insert® Associative Copy® Pattern Feature**



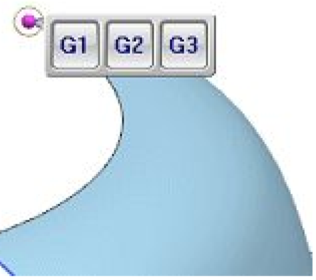
Studio Spline enhancements

What is it?

The **Studio Spline** command has been enhanced in the following ways.

- You can now switch between Through Points and By Poles spline types.
  - o When you switch from Through Points to By Poles, the through points and any internal constraints are deleted.
  - o Only the start and end constraints are retained.

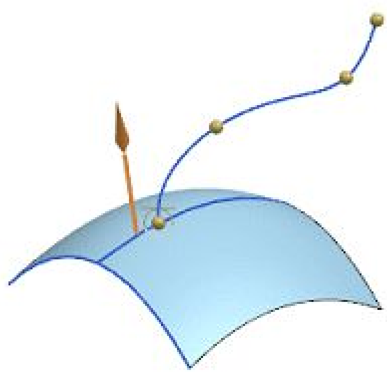
- When you define a By Poles studio spline and reference existing geometry, you can define G1, G2 and G3 constraints for the first spline point immediately after you specify the point.



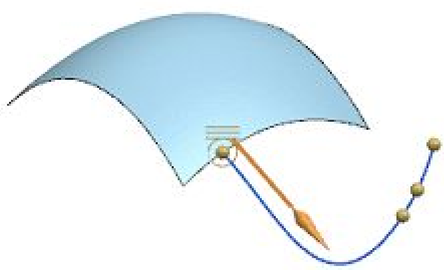
- You can specify degree control for a single segment studio spline. The number of poles is automatically adjusted depending on the degree you specify.

- There are two new inferred constraints available.

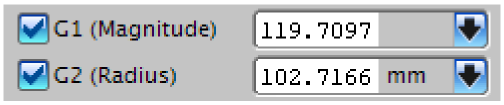
  - o **Normal** – You can infer a G1 constraint from the natural normal of a curve or surface.



  - o **Perpendicular to Curve or Edge** – You can infer G1, G2, and G3 constraints from a curve on surface or surface edge.



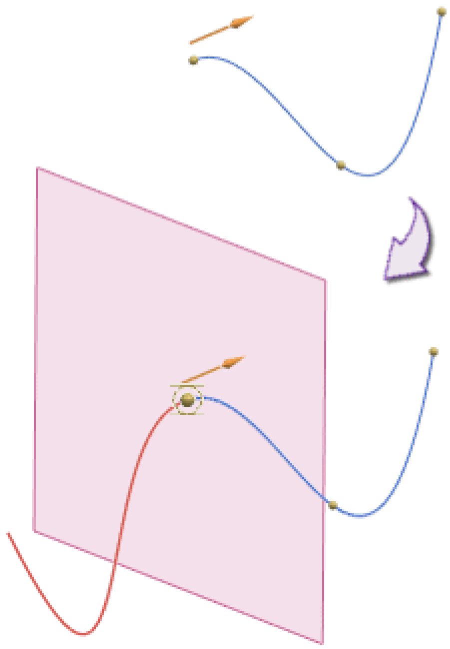
- There are two new methods available for specifying G1 or G2 constraints.



Numeric specification of tangent magnitude

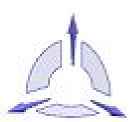
Numeric specification of curvature radius

- When you mirror studio splines, you can symmetrically constrain an end point so both splines are tangent continuous.



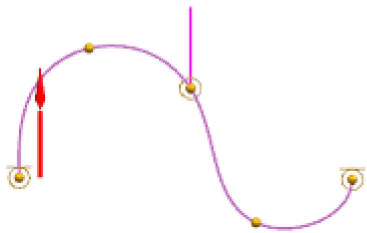
- You can move existing poles or points in the following ways:
  - o Within the view plane.
  - o Along a specified vector.
  - o Within a specified plane.
  - o Along WCS axes or principal WCS planes.
  - o Normal to the spline.

Note

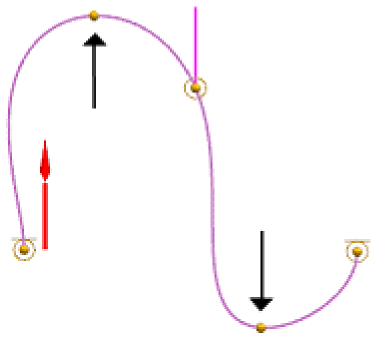


Use the OrientXpress tool to help specify vector and planes during pole and point movement.

- When you move poles and points in the view plane, you can use the Shift key for exact vertical and horizontal movement.
- You can now decide how a spline is updated when parents of its constraints are modified.
  - o You can specify that the spline is updated so all points and poles between the updated point and the next fixed point are moved proportionally.

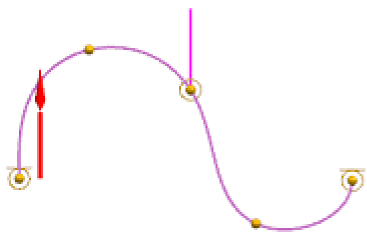


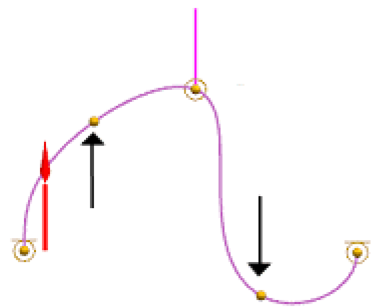
End point of the vertical line is a defining point of the spline.



When the line is moved, the unconstrained points are updated proportionally in relation to the next constrained points.

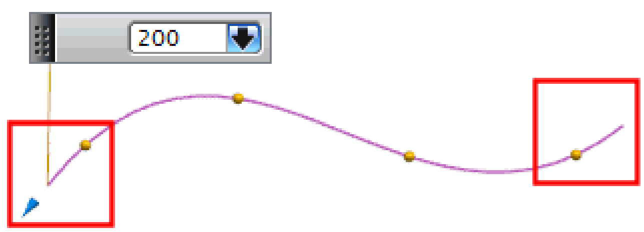
  - o You can specify that only the point of the updated constraint is moved.



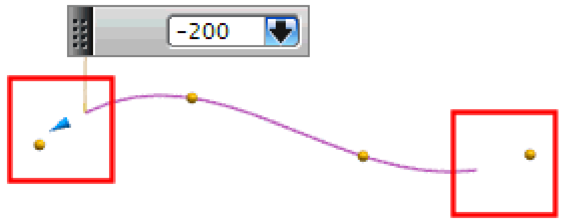


- You can now extend or shorten a studio spline.

Positive value



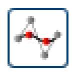
Negative value



- o When you extend a spline, it creates a natural extension on top of the spline; the original spline is not modified.
- o When you extend or shorten a spline, it does not change the endpoint.
- o You can modify the extension during an edit.

- You can use the **Undo** command (Ctrl-Z) to undo any changes to the input spline parameters as long as the dialog box is open and you have not clicked **OK** or **Apply**.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Curve® Studio Spline** 
Menu	**Insert® Curve® Studio Spline**

Copying symbolic threads

What is it?

You can now use **Copy Threads** option when creating copied bodies.

You do not need to re-create symbolic threads that look the same as the symbolic threads on the source body for the following copied bodies in CAM and Drafting:

- Linked bodies
- Extracted-bodies
- Mirrored-bodies

Where do I find it?

Application	Modeling, Shape Studio, Sheet Metal, Manufacturing
Toolbar	**Assemblies® WAVE Geometry Linker** **Feature® Associative Copy Drop-down** list® **Extract Body** **Feature® Associative Copy® Mirror Body** **Feature® Associative Copy® Instance Geometry**
Menu	**Insert® Associative Copy® WAVE Geometry Linker® Body** **Insert® Associative Copy® WAVE Geometry Linker® Mirror Body** **Insert® Associative Copy® Extract Body® Body** **Insert® Associative Copy® Mirror Body** **Insert® Associative Copy® Instance Geometry**
Location in dialog box	**Settings** group® **Copy Threads** check box

Paste Feature enhancements

What is it?

You can now more easily resolve external parent references and understand dependencies when you paste a feature.

- Parent references now appear in the **Paste Feature** dialog box with understandable descriptions.
- You can also see additional information about parent references and their resolution.
- You can resolve dependency references using methods that were available when the feature was originally created.
- You can specify that the dependency geometry be automatically extracted, either associatively or non-associatively.

Why should I use it?

Use **Paste** with the **Paste Feature** dialog box when you want to create a copy of a previously modeled feature.

Where do I find it?

Toolbar	**Standard**® **Paste** ® **Paste Feature** dialog box
Menu	**Edit**® **Paste**® **Paste Feature** dialog box

Advanced Curve Fit

What is it?

The **Advanced Curve Fit** option has been added to the **Project Curve**, **Offset Curve** and **Intersection Curve** commands. This option provides advanced control of the parameterization of the output curves and replaces the **Curve Fit** options of previous releases.

Advanced Curve Fit provides four curve fitting methods:

- **Degree and Segments** – Use this option to specify the degree and segment of output curves. This provides explicit control on the parameterization of output curves

- **Degree and Tolerance** – Use this option to specify the maximum degree and the tolerance to control the parameterization of the output curve.

- **Keep Parameterization** – Use this option to inherit the degree, segments, pole structure and the knot structure from the input curve (or, in the case of **Intersection Curve**, the input surface) and apply it to the output curve.

- **Auto Fit** – This replaces the **Advanced** option of previous releases. Use this option to specify the minimum degree, the maximum degree, the maximum number of segments and tolerance to control the parameterization of the output curve.

  If the output curves do not meet the specified tolerance, then an Alert message is displayed to inform you that the specified tolerance is not met.

The **Align Curve Shape** option has been added to the **Project Curve** and **Offset Curve** dialog box settings.

When selected, **Align Curve Shape** applies the pole distribution of the input curve to the projected or offset curve regardless of the curve fitting method used.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Curve** ® **Project Curve** **Curve** ® **Offset Curve** **Curve** ® **Intersection Curve**
Menu	**Insert** ® **Curve from Curves** ® **Project** **Insert** ® **Curve from Curves** ® **Offset Curve** **Insert** ® **Curve from Bodies** ® **Intersect**
Location in dialog box	**Settings** group

Blend enhancements

What is it?

Both **Face Blend** and **Edge Blend** now support conic blends.

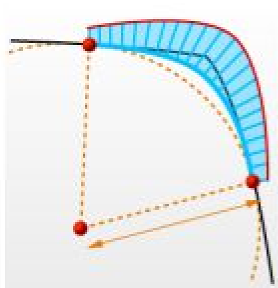
Conic blends have a soft appearance and are used to improve the aesthetic quality and formability of cast and sheet metal parts.

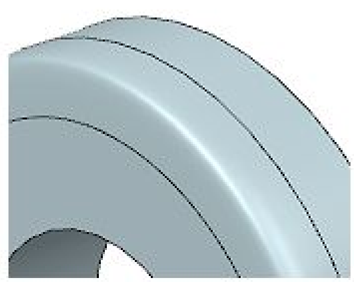
Changes to conics in Face Blend

Face Blend now has two types of conic blends under the **Shape** option menu:

- **Symmetric Conic**

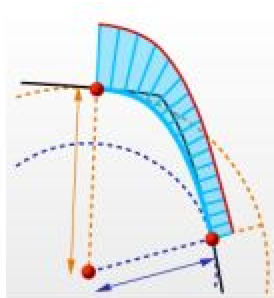
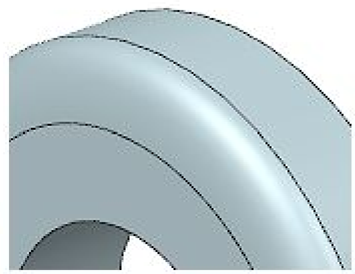
  Employs a single set of handles to simultaneously control both edges of the blend to provide blend symmetry.





- **Asymmetric Conic**

  Uses individual handles to control the edges of the blend, letting you vary the blend on each edge.

Separate conic methods options with their own controls simplify creating the default conic face blends.

Conic shape added to Edge Blend	As with **Face Blend**, **Edge Blend** has a new **Shape** option menu to let you create conic shaped blends.
Conic Method	Conic methods provide choice and flexibility in helping you determine the optimum conic method based on your requirements. You can create the following conic blend shapes: • Boundary and Center • Boundary and Rho • Center and Rho

There are also these additional enhancements to blends.

Trimming a blend	In **Edge Blend**, you can now use an edge or a face as a limiting object. You can reverse the direction of the edge limit type to see alternate solutions for either side of the limit edge. In **Face Blend**, the **Overriding Trim Objects** option is simplified for consistency and improved ease of use.
Stop Short of Corner handles	In **Edge Blend** you can now drag the **Stop Short of Corner** handles across the end of an edge and onto an adjacent edge. This can reduce the occurrence of tiny blends (or slivers) occurring on the blend. This improves its quality and avoids the need to manually construct blend patches to fill any gaps.

Why should I use it?

You can now use some of the more advanced, previously unavailable blending options with simple edge blends.

For both edge and face blends, you can create blends that enhance the aesthetic appearance of your products while also improving their formability and capacity to be manufactured from cast and sheet metal parts.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Feature® Edge Blend** **Feature® Face Blend**
Menu	**Insert® Detail Feature® Edge Blend** **Insert® Detail Feature® Face Blend**

Optimizing curves

What is it?

You can use the new **Optimize Curve** option in the **Modeling Preferences** dialog box to make features more tolerant to change.

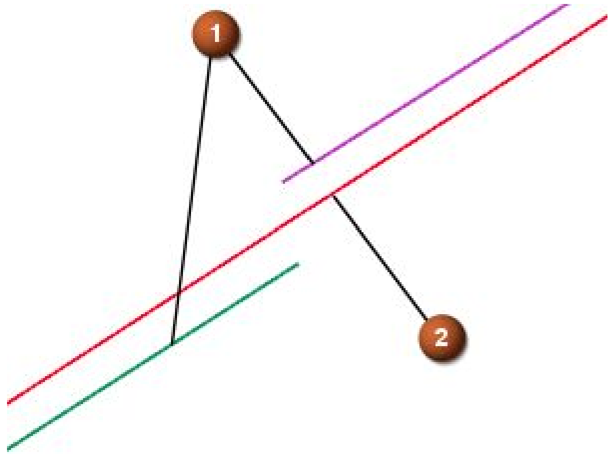
The option helps prevent failures when input curves are changed and the features are updated.

The option introduces an allowed gap size between curves in a selected string by multiplying the current **Distance Tolerance** and **Angle Tolerance** by the **Optimize Curve Tolerance Factor** (below the **Optimize Curve** check box). Any gap or overlap within that size is removed in the output curve when you use the following commands:

- **Ruled**, **Through Curves**, **Through Curve Mesh**, **Sweep Along Guide**, **Swept**, **Section Surface**, and **Law Extension Surface**.

- **Tube**

- **Project Curve**, **Intersection Curve**, **Join Curves**, and **Offset Curve in Face**.



1 Original curves selected for a section string, with gap.

2 Resulting optimized curve used for surface generation (or the projected or intersection curve) using the allowed gap size derived by multiplying the **Distance Tolerance** and the **Optimize Curve Tolerance Factor**.

The **Optimize Curve** check box is cleared by default.

Why should I use it?

You can overcome poorly constructed curves that cause the features using the string of curves to fail when you create or update them.

Where do I find it?

Application	Modeling, Shape Studio, NX Sheet Metal
Prerequisite	In the **Modeling Preferences** dialog box, the **Optimize Curve** check box must be selected.
Menu	**Preferences→Modeling→General→Optimize Curve** **File→Utilities→Customer Defaults® Modeling→General→Tolerances/Scales**.

Changing hole types during edit

What is it?

When you edit a hole of one of the following hole types, you can now change it to any of these other types:

- **General Hole**
- **Drill Size Hole**
- **Screw Clearance Hole**
- **Threaded Hole**

Note When you edit a hole of the **Hole Series** type you cannot change it to any other hole type; nor can you change any other hole type to the **Hole Series** type.

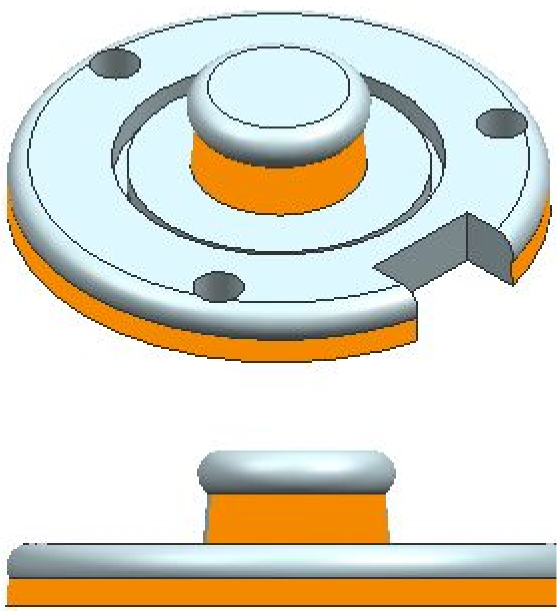
Where do I find it?

Application	Modeling
Toolbar	**Edit Feature® Edit Feature Parameters**
Menu	**Edit® Feature® Edit Parameters** **Edit® Feature® Edit with Rollback**
Graphics window	Double-click a hole feature Right-click a hole feature and choose **Edit Parameters** Right-click a hole feature and choose **Edit with Rollback**
Location in dialog box	**Type** group, **Type** list

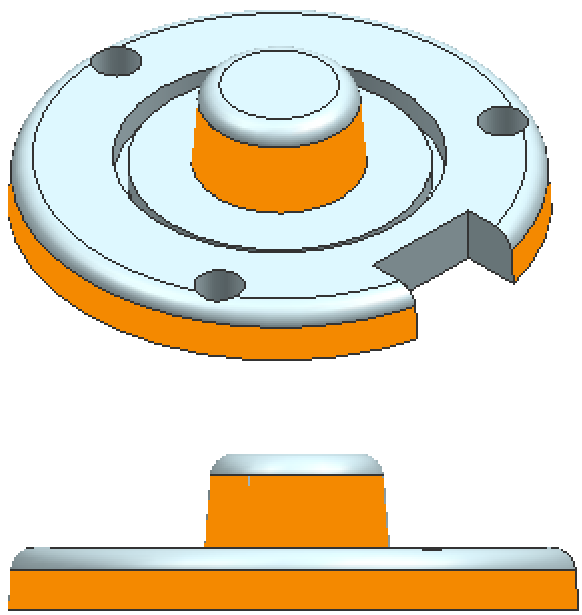
Draft feature enhancement

What is it?

When you add draft to a part that contains blends it is no longer necessary to reorder the blends to be after the draft feature in the Feature Navigator. The blends will remain tangent with the drafted face.



Blends are not tangent.



Blends are now tangent.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Feature® Draft** 
Menu	**Insert® Detail Feature® Draft**

Internationalization of expressions

What is it?

When you create an expression, you can now type the expression name in a supported international language (locale).

DesignLogic functions can accept such localized expression strings as parameters, and return the strings in the same locale.

Why should I use it?

This enhancement is useful if you run NX in languages other than English.

Where do I find it?

Application	Modeling, Shape Studio
Menu	**Tools® Expression**
Location in dialog box	**Name** box **Formula** box

Feature and relations Browser

What is it?

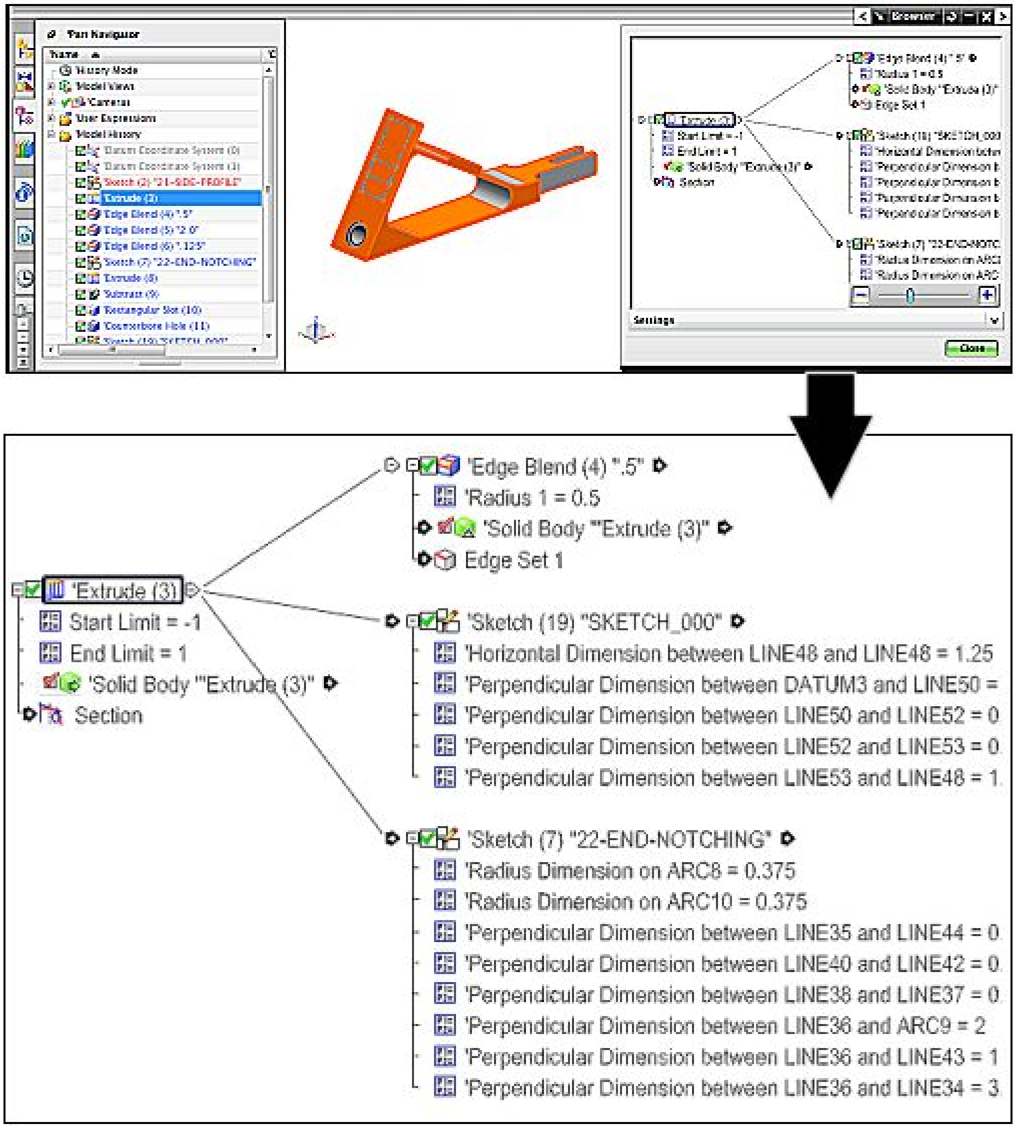
The new **Browser** replaces the former **Feature Browser** with a graphic view of features and their relations.

The browser illustrates how features (and some non-features) are related. You can browse a feature to understand cause and effect:

1. What ancestors affect a feature?

2. If you edit a feature what descendents might be affected?

When you roll over a node in the browser, the respective object highlights in the graphics window and in the **Part Navigator**, either as it exists now or as it was when it was created (depending on the setting of the **Highlight Original** selection preference).



The following types of features appear in the new Browser:

- Body-based features
- Curve features
- Point features
- Datum features
- Sketch features

You can interact with nodes in the browser display.

- To expand a node click the node.
- To display a shortcut menu, right-click a node.
- To edit a node, double-click the node.

You can also browse non-feature relations.

- Expressions (both feature expressions and user expressions)
- Non-feature geometry that has relations to a feature, such as a non-associative curve that is the parent of a feature

You can perform the following actions on geometry in the browser:

- Hide a body (using **Hide Body**)
- Show parents
- Hide parents
- Show a body (using **Show Body**)

You can use the following commands on features in the browser:

- **Suppress Feature / Unsuppress Feature**
- **Replace Feature**
- **Make Current Feature / Make Current Tool Feature / Isolate Tool Body**
- **Delete**
- **Properties**

Why should I use it?

You can browse features in your part and review their relations with other features and objects.

Where do I find it?

Application	Modeling, Shape Studio
Menu	**Information**® **Browser**
Graphics window	Right-click a feature® **Browse**
Part Navigator	Right-click a feature® **Browse** Right-click an expression® **Browse**

Extract Body Here

What is it?

Use **Extract Body Here** to extract a body at a particular time stamp order location.

Why should I use it?

You do not need to use the following commands to extract the body at a particular timestamp: **Make Current Feature**, **Extract** (with the **Fix at Current Timestamp** check box selected), and **Make Current Feature** again.

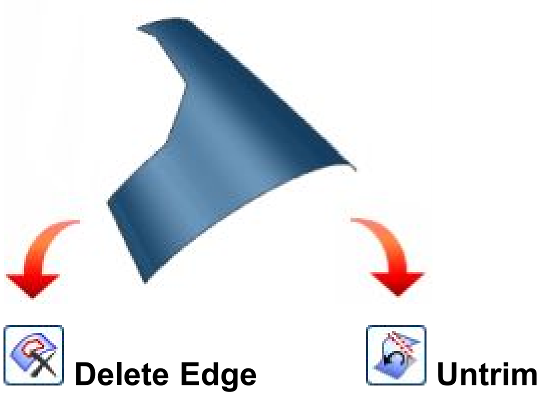
Where do I find it?

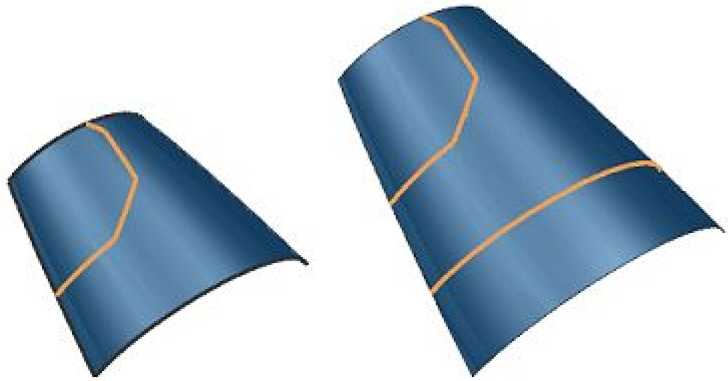
Application	Modeling, Shape Studio, Sheet Metal, Advanced Simulation
Location in the Part Navigator	Right-click a node and choose **Extract Body Here**

Delete Edge

Use the **Delete Edge** command to delete an edge or chain of edges from a sheet body. You can use it to remove a hole, or to untrim an exterior boundary.

If you want to remove specific edges from a sheet body, use the **Delete Edge** command. If you use the **Untrim** command the sheet body is copied. The copy is returned to its natural boundaries and you may need to retrim some edges.





Where do I find it?

Application	Modeling, Shape Studio
Menu	**Insert® Trim® Delete Edge**

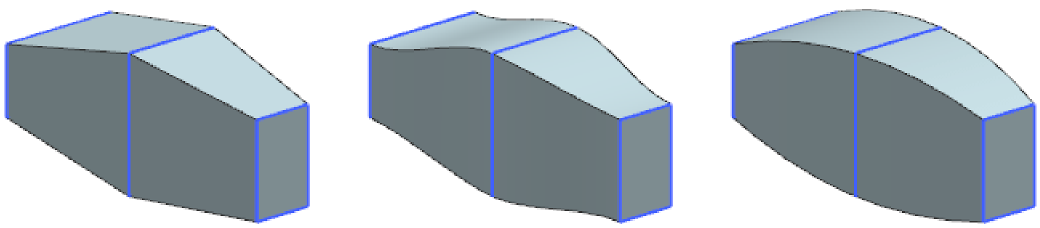
Swept enhancements

What is it?

The **Swept** command has been enhanced in the following ways:

- You can now use the **Blend** interpolation method when sweeping more than two sections along one guide.

  This option has been added to the **Linear** and **Cubic** interpolation methods.



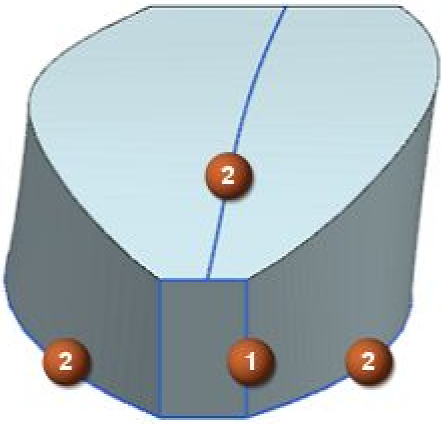
Linear *Cubic* *Blend*

- You can now use the **Another Curve** scaling option when creating a swept feature using two guides.

  o Use the **Another Curve** option to select a curve as a reference for controlling the *height* of a swept surface.

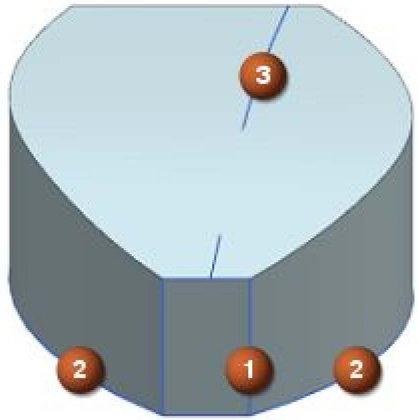
  o As **Another Curve** is a scale curve and not a guide curve, it does not control the surface orientation, as is the case when you use the 3 - guide method. This can help avoid surface distortion problems.

3 Guides



1. *Section rectangle*
2. *Guides (3)*

Another Curve



1. *Section*
2. *Guides (2)*
3. *Another Curve*

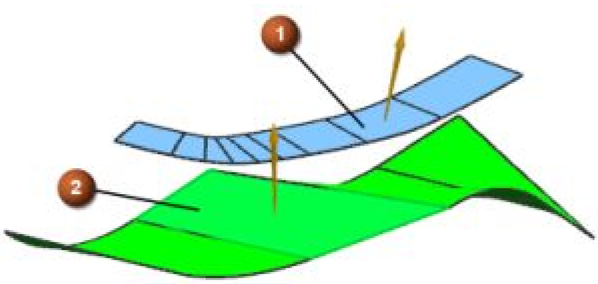
Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Surface® Swept**
Menu	**Insert® Sweep® Swept**
Location in dialog box	**Section Options** group

Editing linked or extracted face/body enhancements

What is it?

When you replace the parents of a linked or extracted object, the direction for the replacement object is now inferred from the replaced object. The directions for both the replaced and the replacement objects are displayed for comparison. You can now determine whether to reverse the direction.



1 Linked seed face being replaced (deselected).

2 Replacement seed face (selected).

Notice both direction arrows are displayed.

Why should I use it?

These enhancements ensure that downstream features do not fail when they are updated.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Edit Feature® Edit Feature Parameters** or **Edit with Rollback**.
Menu	**Edit→Feature→Edit Parameters**, or **Edit with Rollback**.
Graphics window	Double-click an Extracted or Linked feature.

Relief added to threaded holes in a Hole Series

What is it?

You can now add relief to a threaded hole when it is part of a **Hole Series**.

The same **Relief** and **Relief Chamfer** options that are available with the **Threaded Hole** type are now available with threaded holes in the **Hole Series** type.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Feature® Hole**
Menu	**Insert® Design Feature® Hole**
Location in dialog box	**Type** group® **Hole Series** **Specification** group® **End** tab® **Form® Threaded** **Specification** group® **End** tab® **Relief** **Specification** group® **End** tab® **Relief Chamfer**

Variable Offset enhancement

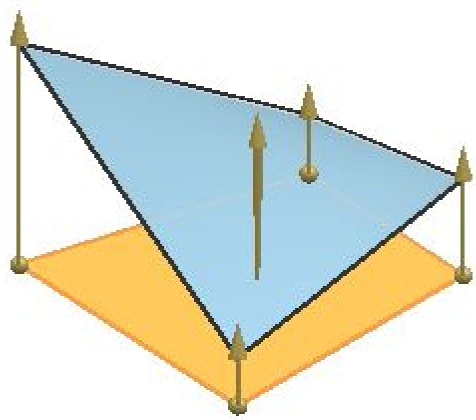
What is it?

When you use the **Variable Offset** command, you now have dynamic control for the offset and location of each offset point.

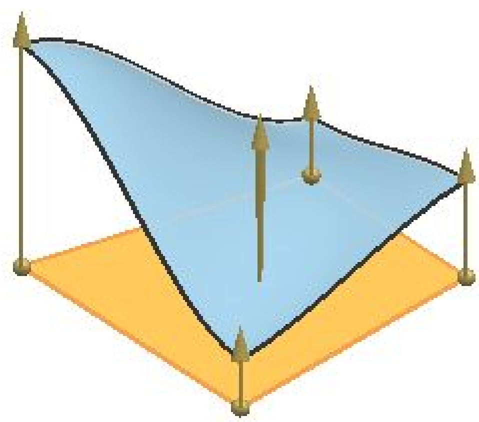
- You can drag a point handle to an offset value or enter an offset value in the on-screen input box or in the dialog box.

- You can drag the point location or enter the point U and V values into the on-screen input box.
- You can select the **Offset** method to be **Linear** or **Cubic**.
- You can use **Keep Parameterization** to maintain the original surface parameters in the Variable Offset surface.
- You can use **Dynamic Deviation display** for visual verification.

In this example, you can see the difference between the offset methods.



Linear Method



Cubic Method

Why should I use it?

You can change the offset values and creation method while creating the **Variable Offset Surface**.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Feature® Offset/Scale Drop-down® Variable Offset** .
Menu	**Insert® Offset/Scale® Variable Offset**

Replace Feature

What is it?

The **Replace Feature** command lets you replace an original feature and its dependents with a replacement feature. The command has the following enhancements:

- When you select a feature, either the original feature or the replacement feature, the new **Add Prior Features of Body** option lets you automatically select ancestor features of the body.

- You can select a dependent of the original feature for the replacement feature. For example, you can replace an original body with an offset of the body.

- When dependent features are changed from an original feature to a replacement feature, only the dependents downstream of the latest original feature will be updated.

Why should I use it?

This enhancement is useful if you model a basic design, add numerous detailed features to it, and then model a new version of the basic design. The enhancements to **Replace Feature** let you replace the original basic design with the new version, while automatically switching the dependent detailed features to the new version.

Where do I find it?

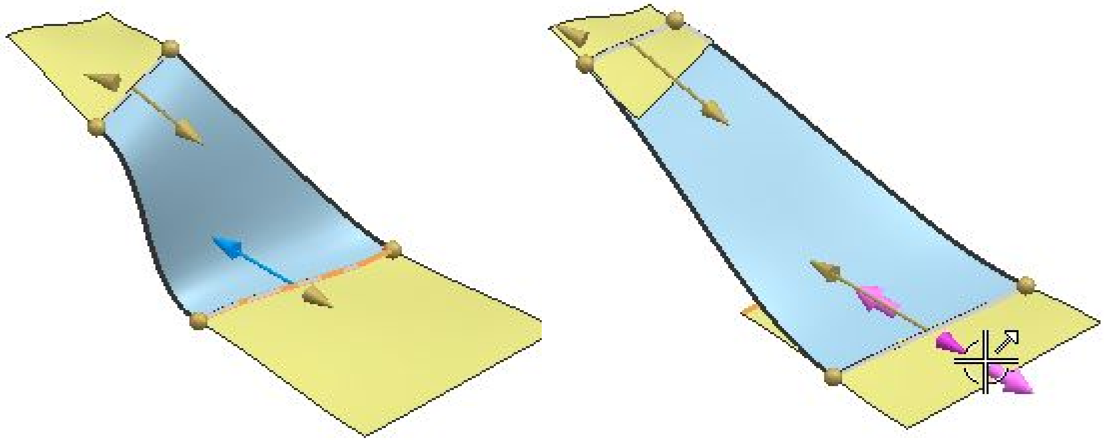
Application	Modeling, Shape Studio
Toolbar	**Edit Feature**® **Replace Feature** 
Menu	**Edit**® **Feature**® **Replace**
Location in dialog box	**Feature to Replace** group® **Related Features**® **Add Prior Features of Body** **Replacement Feature** group® **Related Features**® **Add Prior Features of Body**

Bridge enhancements

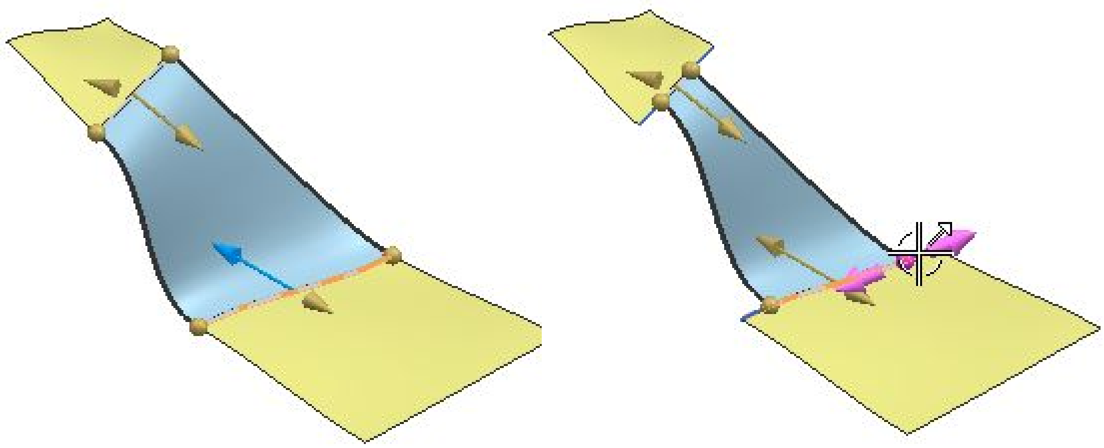
What is it?

When you use the **Bridge** command, you now have more control over the surface parameters.

- You can drag the edges of a bridge surface onto the faces that correspond to the primary input edges.



- You can independently drag the bridge surface vertices to change the edge length. Use this when bridge surfaces are not of even length.



Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Feature® Detail Feature Drop-Down® Bridge**
Menu	**Insert® Detail Feature® Bridge**

Add references to part and object attributes with an expression

What is it?

You can create expressions that reference part or object attributes.

If the part or object attribute is later modified the expression is updated automatically.

Why should I use it?

Use attribute expressions when you need to refer to a part or object as an expression that is dynamically updated.

Where do I find it?

Application	Modeling, Shape Studio
Menu	**Tools® Expression**
Location in dialog box	**Reference Part Attribute** **Reference Object Attribute**

Selection Intent available in commands when specifying points, vectors, planes, and axes

What is it?

Selection Intent is available from within a command for the following option and type settings:

Specify Point option, when the type is set to either of the following:

Point on Curve/Edge

Intersection Point

Specify Vector option, when the type is set to:

On Curve Vector

Specify Plane option, when the type is set to:

Tangent Plane

Why should I use it?

Using Selection Intent with **Datum Plane**, **Datum Axis**, associative **Point**, and other commands that use these option types to specify points, vectors, and planes will enable robust model update by allowing features to adapt to changes to input geometry.

Where do I find it?

Application	Modeling, Shape Studio

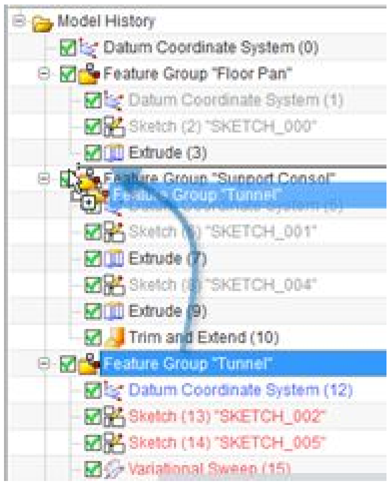
Feature timestamp sequencing

What is it?

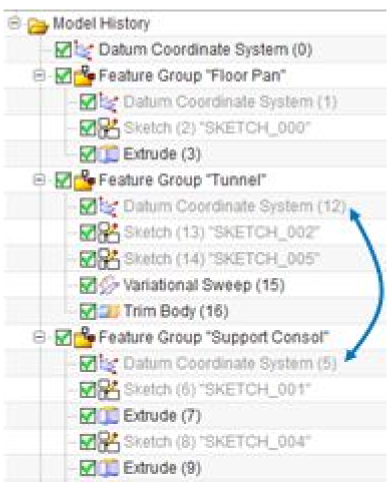
The feature timestamp order is updated in the **Part Navigator** when you use feature groups.

- When you create or reorder feature groups, the member feature timestamps are re-sequenced.
- When you create or reorder feature groups, the gaps in the feature timestamp sequence are removed.

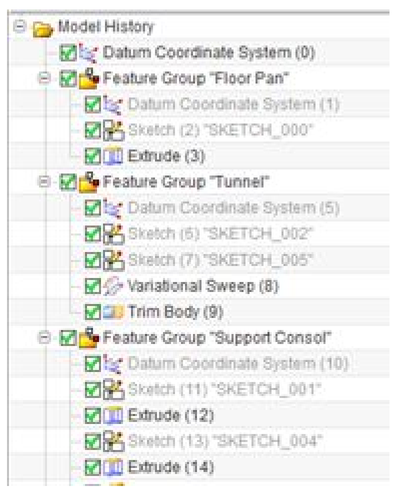
You cannot resequence features of some parent child relationships.



Reorder feature group



Previous result



Updated result

Why should I use it?

- Child features appear in the correct location in the **Part Navigator** with respect to their parents.
- Feature timestamps are generally in order, and the number of missing timestamps are minimized.
- The sequential order of timestamps is not disturbed when you move feature groups, and when you move features into, our of or between feature groups.

Where do I find it?

Application	Modeling and Shape Studio

Extension enhancements

What is it?

The **Extension Surface** command options are redesigned based on typical surface extension workflows and the dialog box has been updated.

Type

- You can specify **Edge** or **Corner** as the extension type.

Edges to Extend

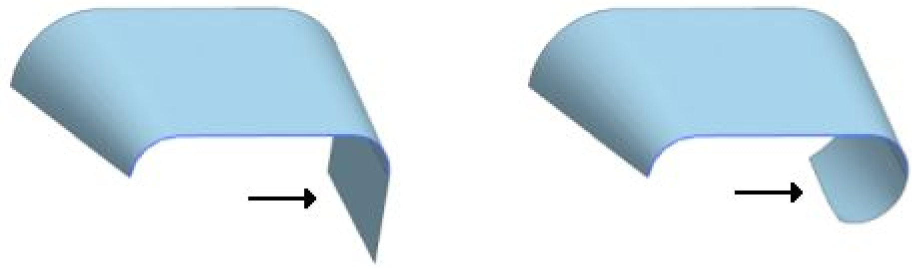
- When you use the **Edge** type, you can specify the edges to be extended.

Corner to Extend

- When you use the **Corner** type, you indicate a corner of a face to be extended.

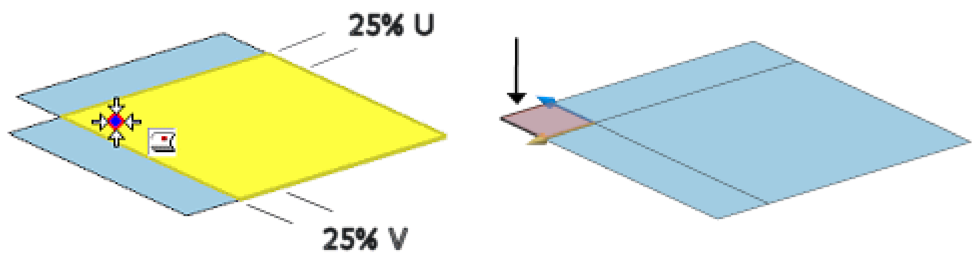
Extension

- For the **Edge** type, you can specify either a tangential or circular method of extension and specify the distance of the extension either by length or a percentage of the original face.



Tangential example **Circular example**

- For the **Corner** type, you can specify a percentage of the original face in the U and V directions to specify the size of the corner extension.



Note The **Angled** and **Normal to Surface** options that were available in previous releases have been removed because similar functionality is available in the **Law Extension** command.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Surface® Extension Surface**
Menu	**Insert® Flange Surface® Extension**

Divide Curve Enhancement

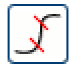
What is it?

You can now divide curves using multiple bounding objects, when you set **Type** to **By Bounding Objects**.

Why should I use it?

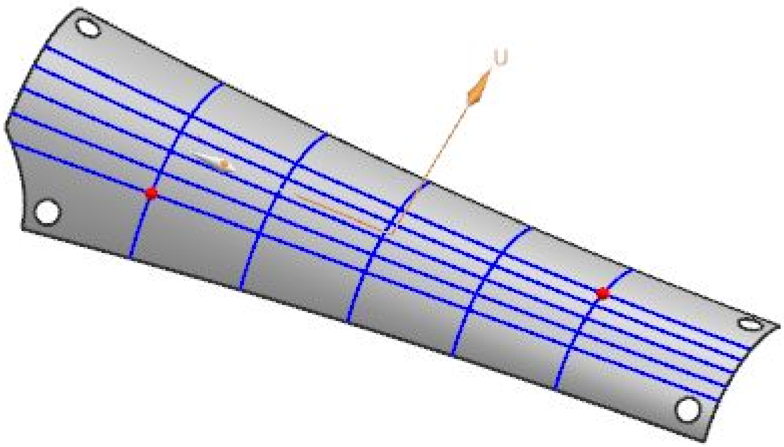
This enhancement lets you get multiple divide curve results using bounding objects when you use the **Divide Curve** command.

Where do I find it?

Application	Modeling
Toolbar	**Edit Curve® Divide Curve** 
Menu	**Edit® Curve® Divide**

Isoparametric Curve

You can create feature-based isoparametric curves from surfaces.



Isoparametric curves can be useful in constructing adjacent surfaces that follow the flow lines of the parent surface.

Since Isoparametric curves are features they are easily edited and updated.

Note This command replaces the former **Isoparametric Curve** option found under the **Extract Curve** command.

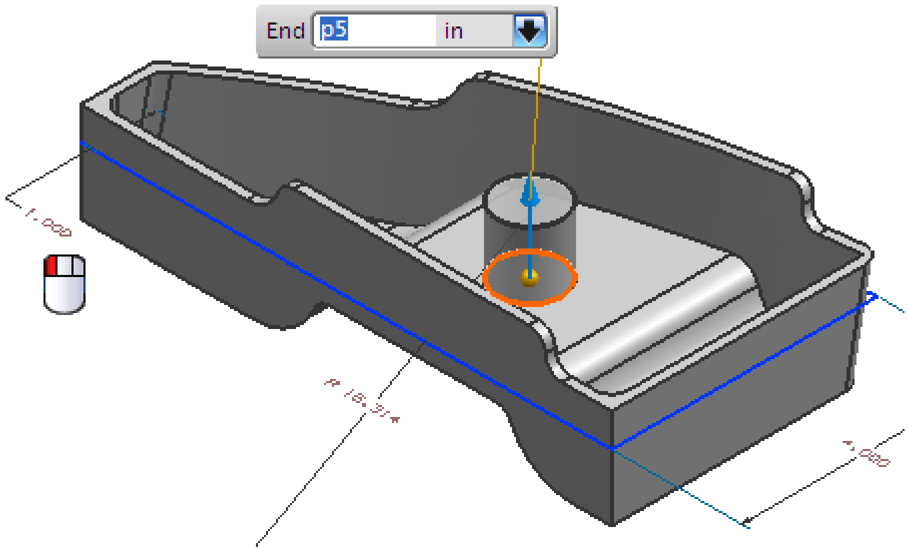
Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Curve® Isoparametric Curve**
Menu	**Insert® Curve from Bodies® Isoparametric Curve**

Referencing feature dimensions enhancement

What is it?

You can now select feature dimensions, sketch dimensions, and auto dimensions from the graphics window when you use the DesignLogic Reference option.



Why should I use it?

You can quickly select the dimensions from the graphics window instead of selecting them from a list box.

Where do I find it?

Application	Modeling
Location in dialog box	[parameter entry] **DesignLogic** list® **Reference**.

Selecting objects from the Part Navigator

What is it?

You can select the following nodes in the **Part Navigator** as input to the active command:

- Body features as body input
- Curve features as curve input, when the **Feature Curves** selection intent rule is available
- Body features as face input, when the **Feature Faces** selection intent rule is available
- Body objects as face input, when the **Body Faces** selection intent rule is available

When **Select Object** is the active option in a command, you can select the following objects in the **Part Navigator**:

- Point features as point input, when **Point** is available on the Selection bar **Type** filter.

- Datum CSYS features as CSYS input, when **CSYS** is available on the Selection bar **Type** filter.

Why should I use it?

In a complicated part, it can be easier to select items from the **Part Navigator** instead of the graphics window.

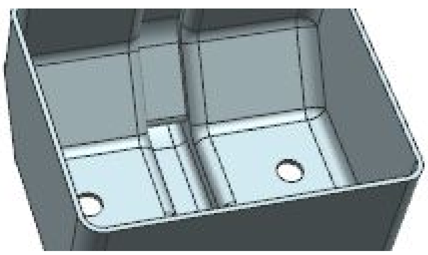
Where do I find it?

Application	Modeling, Shape Studio
Menu	**Tools® Part Navigator**

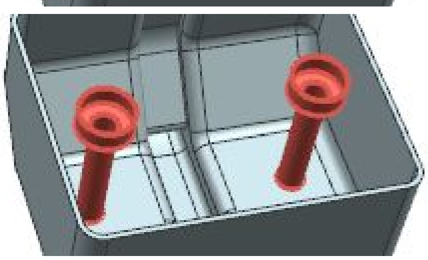
Display features as created

What is it?

When a feature is selected or preselected, the feature now displays as it was when it was created.



Complete part



Holes selected, revealing the counterbored hole faces that have since been consumed (deleted by subsequent features).

Why should I use it?

It is automatic. This is the way features now display.

Where do I find it?

Prerequisite	A feature must be selected or pre-selected, either in the graphics window or in the Part Navigator

Law Extension – Keep Parameterization

What is it?

The **Law Extension** command has updated **Rebuild** options.

- **Degree and Tolerance** – Use this option to specify the maximum degree and the tolerance to control the parameterization of the output surfaces.

- **Auto Fit** – This replaces the Advanced option of previous releases. Use this option to specify the minimum and maximum degree, maximum segments and tolerance to control the parameterization of the of the desired output surface.

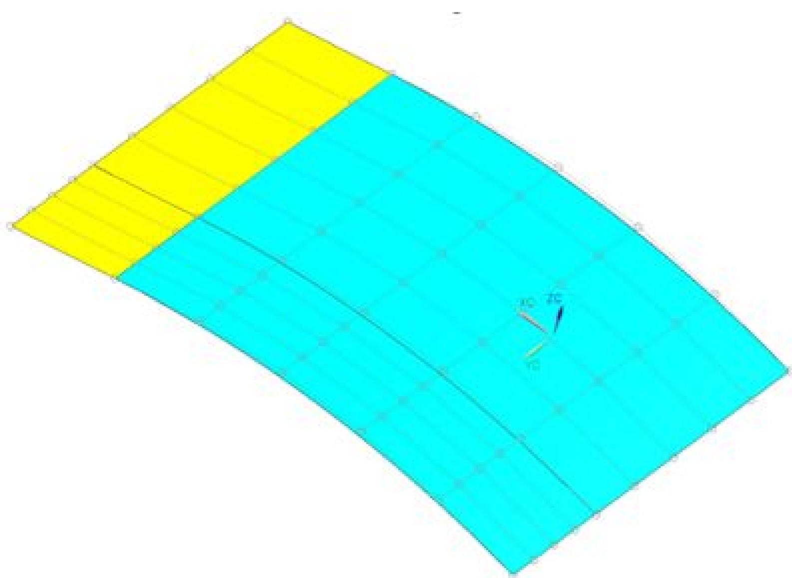
  If the output surfaces do not meet the specified tolerance, then an Alerts message will be displayed to inform you that the specified tolerance is not met.

- **Keep Parameterization** – Use this option to inherit the degree, segments, pole structure and the knot structure from the input surfaces and apply it to the output surface.

  - o G0/G1 tolerances will not be considered.

  - o Each law extension surface will have the same degree and segments as its input curve.

  - o The continuity between adjacent law extension surfaces will follow the continuity of their corresponding curves.



Law extension surfaces are at the upper left

In addition, both law extension surfaces and studio surfaces are now created in a more streamlined manner.

- Surfaces are produced with the least number of poles necessary to create the surfaces so it is easier to edit the surfaces.
- Surfaces are created with no C0 or C1 knots.
- Surfaces are created with reasonable knot spacing and regular pole structure, with spacing between poles and rows that produce minimal inflections.

Why should I use it?

Use the **Keep Parameterization** option to create surfaces with the same parameterization of the input curves, ensuring subsequent surface operations will be of high quality.

Also, keeping parameterization between input curves and derived surface makes it easier to go between a mesh and sweep based workflow and a pole editing based workflow.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Surface ® Law Extension** 
Menu	**Insert ® Flange Surface ® Law Extension**
Location in dialog box	**Settings** group

DesignLogic List expressions

A new expression **List** data type lets you streamline NX DesignLogic interactions and provide additional functionality capable of handling a wider variety of design tasks.

With the **List** data type you can:

- Store and access arrays using the expression sub system.
- Access arrays using lists related DesignLogic functions.

Expressions of the list data type are called *list expressions*. You can use the **Extended Text Entry** option to conveniently specify list expressions with comma separated values of any DesignLogic data type as list expressions using braces {}, as shown below:

coordinates={1,1.5,2}

Why should I use it?

Use these enhancements to operate on an array of expressions or values of different data types.

Where do I find it?

Application	Modeling
Menu	**Tools® Expression® Extended Text Entry** **Tools® Expression® Extended Text Entry® Insert Function** **Tools® Expression® Extended Text Entry® Insert Conditional**

Durability analysis in the Modeling application

What is it?

The **Durability Wizard** is now available from the Modeling application. To run the **Durability Wizard**, you need to have a Simulation file that contains stress or strain results from a static stress analysis solution. To obtain this Simulation file, use the **Stress Wizard**.

Why should I use it?

You can test your parts for fatigue damage directly in the Modeling application by doing a durability analysis.

Where do I find it?

Durability Wizard

Application	Modeling, Design Simulation, Advanced Simulation
Prerequisite	Results from a static stress analysis solution
Menu	**Tools ® Durability Wizard**

Stress Wizard

Application	Modeling, Design Simulation, Advanced Simulation
Resource bar	**Process Studio ® NX CAE Stress Wizard**

Assemblies

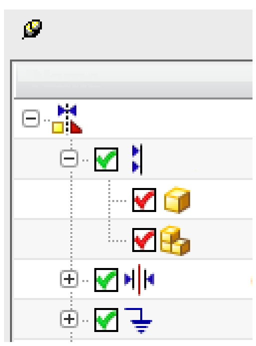
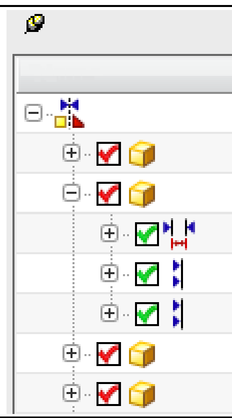
Constraint Navigator

What is it?

Use the **Constraint Navigator** to analyze, organize, and work on assembly constraints in your work part.

As in the **Assembly Navigator**, the **Constraint Navigator** has columns that you can display, hide, and reorder. Some columns are specific to assembly constraints, such as **Constraint Type** and **Creation Date**. To access the column options, right-click the title bar and move your cursor over **Columns**.

You can also group the navigator tree nodes in different ways to analyze the information you are interested in. For example, the **Group by Components** option makes it easy to see which constraints are used to position each component, and the **Group by Constraints** option makes it easy to find all failing constraints. Right-click the title bar and choose the grouping method that you want. The following figure compares two grouping methods for the same assembly.

	
Group by Constraints	**Group by Components**

You can group the navigator tree nodes by any of the following:

- Constraints
- Components
- Status
- Component level
- Inherited
- Component status

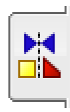
Why should I use it?

The **Constraint Navigator** makes it easy for you to do the following:

- Find and work on constraints that are of interest to you.

- Resolve constraint issues.

Where do I find it?

Resource bar	**Constraint Navigator** 

Unique part files from existing components

What is it?

Use the **Make Unique** command to create a new part file for one or more selected occurrences of the same part.

For example, the following figure shows four selected occurrences of a part named **GKballjoint**. You can use the **Make Unique** command to convert the two occurrences near the tires to use a new unique part file named **GKballjoint_tire**, which is created by this command as a copy of **GKballjoint**. The other two occurrences, which are attached to the steering column subassembly, continue to reference the **GKballjoint** part file.

Why should I use it?

When you create a unique part file, you can modify the occurrences that reference the new part without affecting occurrences that still reference the original part. For example, you can modify the **GKballjoint_tire** part file so its occurrences fit larger tire subassemblies without affecting the **GKballjoint** ball joints that are attached to the steering column subassembly.

Where do I find it?

Application	Assemblies
Menu	**Assemblies® Components® Make Unique**
Graphics window	Right-click a component® **Make Unique**

Load Interpart Data enhancements

What is it?

The **Load Interpart Data** assembly load option, which you can select to ensure that sufficient data from components is loaded to enable interpart updates, has the following enhancements:

- Loading of component data to enable interpart relationships to update is quicker and more memory-efficient than in previous releases. NX is more selective about when to load extra data in order to support interpart updates. These improvements also apply to the **Assemblies® WAVE® Load Interpart Data** command.

- When you make changes in your assembly that cause interpart relationships to become out-of-date, NX loads extra data from partially-loaded components as needed to update the relationships. Previously, NX only applied the **Load Interpart Data** assembly load option when you loaded new components.

Examples of interpart relationships include:

- Interpart expressions
- WAVE links
- Promotions
- Assembly constraints

Why should I use it?

The ability to dynamically load extra data from partially-loaded components whenever interpart relationships become out-of-date makes it easier to keep your assembly model up-to-date.

To optimize the performance of assembly loading without losing the ability to update interpart relationships, you should use the **Load Interpart Data** assembly load option in conjunction with partial loading.

Where do I find it?

Application	Assemblies
Menu	**File**® **Options**® **Assembly Load Options**
Location in dialog box	**Scope** group® **Load Interpart Data**

Load Interpart Data command

Application	Assemblies
Menu	**Assemblies**® **WAVE**® **Load Interpart Data**

Reopening modified parts

What is it?

When you select one or more component nodes in the **Assembly Navigator**, you can use the following new shortcut commands:

- Use the **Reopen Part** command to close selected components that are modified and reopen their saved versions. A component is regarded as modified when it has changed in your NX session, or when the underlying part file in Teamcenter or the operating system has been updated.

- Use the **Reopen Assembly** command to close modified part files in a subassembly and reopen those part files. A subassembly includes the selected part file and all its children.

When you select a single component node, the commands appear only if the component is modified.

Why should I use it?

When you are working in the **Assembly Navigator**, the **Reopen Part** and **Reopen Assembly** commands provide a quick way to discard unwanted modifications to a part or to refresh a loaded part with an updated version from Teamcenter or the operating system. This is faster to use than the **Reopen Part** dialog box options.

Where do I find it?

Application	Assemblies
Assembly Navigator	Right-click one or more selected component nodes® **Close**® **Reopen Part** or **Reopen Assembly** Right-click the **Component Groups** node® **Components**® **Close**® **Reopen Part** or **Reopen Assembly**

Edit Suppression State enhancement

What is it?

When you use the **Edit Suppression State** command, you can now select multiple components in different branches of your assembly structure and change the suppression state at any level from the lowest common parent to the top-level assembly.

Why should I use it?

Previously, you could select multiple components only when they were all children of the same occurrence.

Where do I find it?

Application	**Assemblies**
Menu	**Assemblies**® **Components**® **Edit Suppression State**
Assembly Navigator	Right-click a component node® **Suppression**
Location in dialog box	**Controlling Parent** list box

Advanced Weight Management enhancements

What is it?

The **Advanced Weight Management** command is enhanced to improve support for component groups. If you specify a component group to define which components to include in an assembly weight calculation, NX re-evaluates the component group before making the calculation to ensure that it includes the components that currently satisfy the component group criteria.

Options on the **Weight Management** dialog box that previously referred to a component set now refer to a component group. For example, the **Set Component Set** option is now the **Set Component Group** option.

Where do I find it?

Menu	**Analysis® Advanced Mass Properties® Advanced Weight Management**

Read-only work parts

What is it?

When you attempt to make a read-only part the work part, a message informs you that the part is read-only. You can control the display of the message by using the **Display Message when Work Part is Read-only** customer default.

Why should I use it?

The message makes it more obvious when your work part is read-only. Without the message, you might not realize that your work part is read-only until you attempt to save the work part after making modifications.

Where do I find it?

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Assemblies® General® Miscellaneous**

Synchronize Links

What is it?

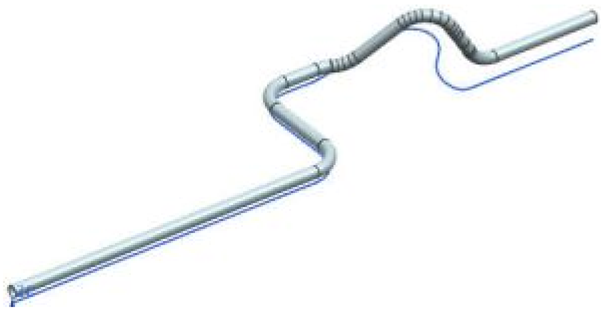
The following enhancements help you update a linked feature and its dependent features to match changes in the source feature:

- A **Synchronize Links** command
- A **Reverse Direction** option in the **WAVE Geometry Linker** dialog box when **Type** is set to **Datum**

Use the **Synchronize Links** command to do the following:

- Update an out-of-date linked feature to the current version of its source.
- Map descendents of the linked feature to changes in the linked feature in order to update the descendents.

Mapping cross-references topology objects in the out-of-date geometry to the current version of the source, so that descendents that reference the topology are adopted by their new parent. Topology objects include faces, edges, and curves.



Why should I use it?

In a WAVE link, it is not uncommon to delay updating in the linked feature. The **Synchronize Links** command provides the ability to not only update the linked feature, but to map all of its dependent features.

Where do I find it?

Menu	**Assemblies® WAVE® Synchronize Links**

Assembly constraints icons

What is it?

Beginning in NX 7.5.2, in the **Assembly Navigator**, unsolved assembly constraints are marked with new icons that indicate the severity and nature of the situation. The severity icons are error, warning, and info. The category indicator identifies the specific problem. The same icons appear in the **Info** column, where they have tool tips that describe the problem.

Icons also appear on the **Constraints** folder and its **Info** column when any of its constraints have a severity status, are out-of-date due to constraint delays, or are being ignored in the current arrangement.

Note In NX 8.0, a new navigator (the **Constraint Navigator**) is being introduced. The icon changes described in this topic also apply to the **Constraint Navigator**, except that additional information appears in the **Status** column instead of the **Info** column.

For example, suppose a constraint has an error because its components cannot be moved relative to each other.

- In the left-most **Assembly Navigator** column, a icon appears beside the constraint name. The same icon appears in the **Info** column with a tool tip that says `Referenced objects cannot move relative to each other.`

  Note If you need additional information, you can right-click an unsolved constraint and choose **Information**. For this example, the **Information** window says `The objects referenced by this constraint cannot move relative to one another.  This may be due to the presence of Fix or Bond constraints.`

- The **Constraints** folder that contains the error has a icon. The same icon appears in the **Info** column, where its tool tip says `Some constraints have errors.`

  Note If your current assembly arrangement is using the **Ignore All Constraints** arrangements property, the **Constraints** folder has a icon instead with a tool tip that says `All constraints are ignored in the current arrangement.`

See the *Assemblies* Help for more information.

Why should I use it?

Icons now clearly indicate the severity and category of the problem for each unsolved constraint. You also have faster access to additional information by using the new **Info** column tool tips.

Where do I find it?

Application	Assemblies
Resource bar	**Assembly Navigator** **Constraint Navigator**

Fix and Bond constraints

What is it?

The ability to drag components is disabled during the creation or editing of Fix or Bond assembly constraints.

Why should I use it?

Because Fix and Bond constraints are designed to hold components in place, this functionality was disabled to prevent any accidental movement of the components.

Where do I find it?

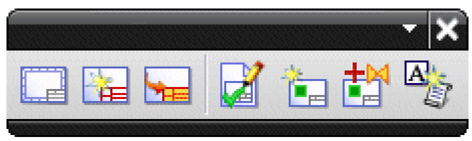
Application	Assemblies
Toolbar	**Assemblies® Assembly Constraints**
Menu	**Assemblies® Component Position® Assembly Constraints**
Location in dialog box	**Type® Fix** or **Bond**

Drafting

DraftingPlus enhancements

Drawing templates

A set of commands on the **Drawing Format** toolbar let you create and edit your own custom drawing templates.



Drawing Format toolbar with all buttons displayed

You can:

- Create and edit associative borders and zones for each drawing sheet in the template file.
- Construct and modify a custom title block from one or more tabular notes.
- Mark the current drafting part as a reusable drawing template.
- Create and link template regions.
- Associate notes, tables, symbols and views with a template region.
- Apply Knowledge Fusion rules to govern the behavior of objects in the template when it is inserted into another part.

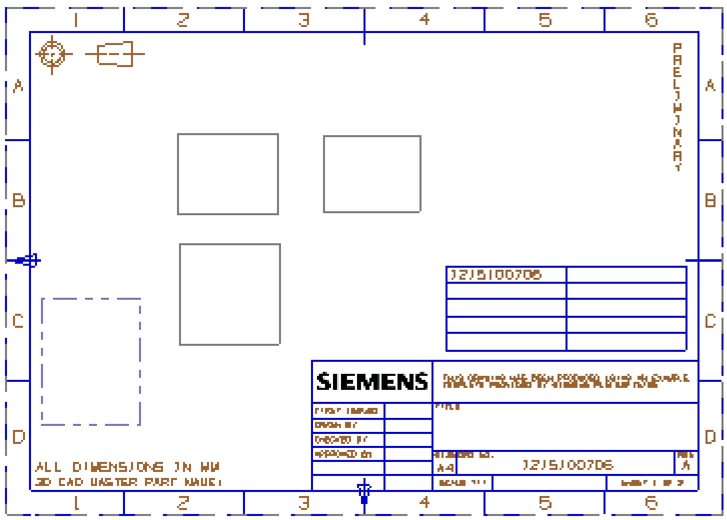
Where do I find it?

Application	Drafting
Toolbar	**Drawing Format**
Menu	**Tools® Drawing Format**

Mark as Template

The **Mark as Template** command creates a sheet or drawing template based on the configuration of the currently open drawing sheet. The template can include associative borders and zones, views, notes, template regions, symbols, and custom title blocks.

Marking a drawing as a template gives you access to the template regions and rules functions as well as allows you to choose to add the template to your collection of sheet and drawing templates.



If you elect to save the template in your template directory, you must specify a presentation name, description, template type, and the .pax file location.

Three different template types are available:

Sheet

Creates a sheet template from the current drafting part. A sheet template is used to add a new sheet to an existing drawing.

Reference Existing Part

Creates a master model drawing template from the current drafting part. A master model drawing template is used to create a separate drafting part that includes the current model as a component.

Standalone

Creates a 2D drawing template. A 2D template is used to create a standalone drawing that does not reference a master model part.

When you save the template part, the specified .pax file is created or updated and the template is displayed by its presentation name on the Resource bar palette.

Note You must have write permission to the .pax file and file directory to add a new template or update an existing template.

Where do I find it?

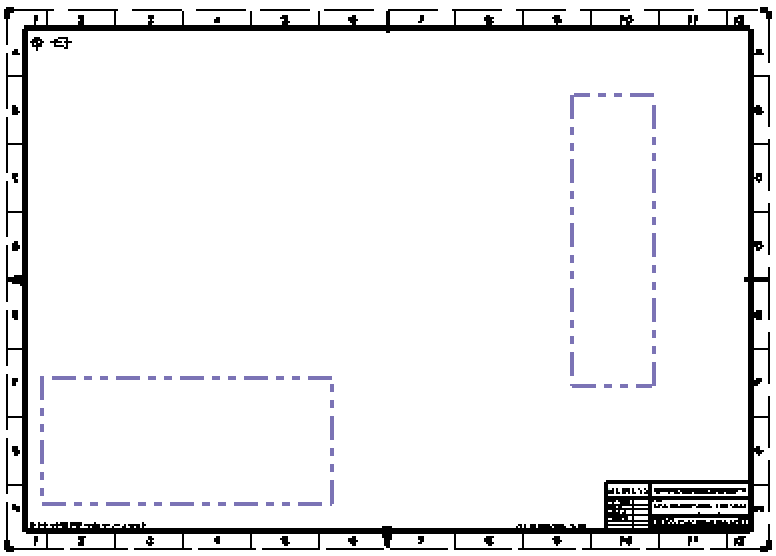
Application	Drafting
Toolbar	**Drawing Format→Mark as Template**
Menu	**Tools→Drawing Format→Mark as Template**

Template Region

A template region is a rectangular area on a drawing template designated to accept only certain types of drafting objects. The **Template Region** command defines the origin and the rectangular dimensions of the area on the drawing template and the specific types of drafting objects that can be contained therein.

You can designate a template region as a note, symbol, table, or view type region. This designation determines the type of drafting object that can be added to the template region. You can apply Knowledge Fusion rules to template regions to govern the behavior of objects in them when the drawing template is incorporated into another part.

Note Although this command is only available for drawing templates, it is not necessary to create template regions in order to add drafting objects to the drawing template. You can add views, custom symbols, title blocks, and notes to your drawing template without template regions.



Phantom lines denoting the boundary of a template region

When the contents of a template region grow beyond its borders, the region behaves in one of two ways:

As a growing region

Contents that extend beyond the template region’s boundary continue into other template regions of the same object type. Depending on how you set the Continuation for the template region, the overflow extends to other template regions of the same type on the same sheet, or to template regions of the same type on another sheet.

As a fixed region

Contents that extend beyond the template region are displayed outside the boundary of the region and do not continue to any other region.

Note View regions are always fixed regions.

After you create a template region, you can re-size it, move it, or delete it in the drawing template.

However, after you import a region as part of a drawing template, you cannot edit it, move it, or delete it.

Priority and Content Control

If content from two growing regions grow into each other, the region with the larger priority number will move to the continuation area. You may also control whether the entire contents of the lower priority region move to another region, or if only the overlapping portion moves.

Customer Defaults

Customer defaults govern the display of template regions. The defaults are contained in the **Customer Defaults** dialog box, on the **Drafting**® **Drawing Automation** ® **General** tab:

Display Region in Non-Template Part — Displays the template region’s border when a drawing template is imported into another part.

Display Region Label — Displays the template region’s name, type, and the rule file name at the center of each region.

You can also set color, font, and width settings for region borders here.

These options also control the default setting for the template region options on the **General** tab of the **Drafting Automation Preferences** dialog box.

Where do I find it?

Application	Drafting
Prerequisite	You must be in a drawing template, which was created using the **Mark as Template** command, to make this command available.
Toolbar	**Drawing Format→Template Region**
Menu	**Tools→Drawing Format→Template Region**

Adding objects to a region

After a drawing has been marked as a drawing template and template regions are added to it, you can add objects like notes, symbols, tables, or views to the regions in your drawing template.

Note When you add an object to a template region, it must be compatible with the region's type and must lie within the region's boundary. NX will not automatically relocate the object for you. When you add an object to a template region, any rule associated with the region is applied to that object when the drawing template is used to generate a drawing sheet.

You can add objects to a template region using the following methods:

To	Do this
Add custom symbols from the Reuse Library to a template region.	Use the **Add Symbols to Region** command.
Add an existing table to a template region.	Right-click the table and choose **Add to Region**. Only one table can be added to a template region.
Add an existing note to a template region.	Right-click the note and choose **Add to Region**. Multiple notes may be added to a template region.
Add an existing view to a template region.	Right-click the view border and choose **Add to Region**. Only one view can be added to a template region.
Add an existing symbol to a template region.	Right-click the symbol and choose **Add to Region**. Multiple symbols can be added to a template region.

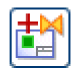
To control the appearance and position of objects in a template region. For example, to control the orientation and scale of views, the application of view labels, along with the location and behavior of tables via the TableData and TableDataFromFile.	Use the **Select Rule** option to assign specific Knowledge Fusion rules to the template region.

The commands becomes available when the drawing is marked as a template.

To remove objects from a template region, right-click the region boundary and choose **Remove Objects from Region**.

Where do I find it?

Add Symbols to Region command:

Application	Drafting
Prerequisite	You must be in a drawing template, which was created using the **Mark as Template** command, to make this command available.
Toolbar	**Drawing Format→Add Symbols to Region** 
Menu	**Tools→Drawing Format→Add Symbols to Region**

Add to Region command:

Application	Drafting
Graphics window	Right-click a drafting object (view, table, and so on)® **Add to Region**

Remove Objects from Region command:

Application	Drafting
Graphics window	Right-click a template region's border® **Remove Objects from Region**, and then select the object to remove.

Automation Rules

The **Automation Rules** command adds configurable Knowledge Fusion-based rules to your drawing template. You can use automation rules to define the behavior and location of automatic dimensions, symbols, and notes when the template is used to create a drawing.

The Knowledge Fusion rules are the mechanism by which information is passed to the drawing file. The actual rules may be written in any programming language such as .NET, Visual Basic, C, C+, or NX/Open.

Drafting Automation Preferences

When you apply automated dimensioning rules to a template part, you can control dimension placement and behavior with the **Drafting Automation Preferences** dialog box. For example, on the **Annotation Distribution Rules** tab, you can specify the minimum and maximum distances between dimensions and geometry, as well as minimum distances between adjacent annotations. You can also prioritize the application of the rules. The preferences become effective when you add the template to a part.

Where do I find it?

Application	Drafting
Prerequisite	This command becomes available only after the drawing is marked as a template.
Toolbar	**Drawing Format→Automation Rules**
Menu	**Tools→Drawing Format→Automation Rules**
Preferences	**Preferences→Drafting Automation**
Customer Defaults	**Drafting→Drawing Automation**

Automated drawing customization

The automation tools and processes available in NX Drafting lend themselves to a great deal of user customization. A lot of basic customizations can be managed with customer defaults. More extensive customizations can be managed with a system of automation rules.

There are two types of automation rules available:

Simple – These rules do not require mathematical or complex logic and can be configured using customer defaults, for example, Annotation Distribution and Dimension rules.

Knowledge Fusion (**KF**) – These rules require more complex logic, such as configuring drawing templates and booklets. Although they are constructed as a KF routine, they can be used to directly call a C, C++, or .net program from the NX Open Application Programming Interface (API).

For example, a KF rule can be used to assign the default orientation of a base view and the application of view scales and labels in a drawing template. The rules are saved with the template part and evaluated when the template is used to create drawings. For booklets, KF rules define the booklet names, booklet disciplines, types, and available templates.

A repository of KF rules is found in the *UGII_BASE_DIR/draftingplus/automation/example* directory. You can copy any of them to a writable location and customize them to your specifications.

Applying rules to drawing templates

You can imbed rules into custom templates to automate dimension, note, and symbol creation when the template is used to construct a drawing in another part.

The KF rule applied to a template region affects the behavior of drafting object(s) associated with that region. A few of the examples you can utilize are:

Template Region Type	KF file
View	*ViewScale.dfa* Defines the scale (1:1) of the view added to the template region. *ViewLabelText.dfa* Defines the label content of a view.
Table	*SampleReferenceTable.dfa* Supplies table data to create and populate a tabular note when the template is used to create a drawing.
Note	*NoteBuilderExample.dfa* Adds text to the note region.
Symbol	*SymbolDataExample.dfa* Adds symbol data to a symbol region.

Tip You can also apply a rule directly to an annotation or symbol that appears on the drawing by right-clicking the annotation or symbol and selecting **Specify Rule**. Only those rules available for that object type will be selectable.

You can also manually create and position rules-based dimensions to drawing views with the **Annotate Views** and **Distribute Annotations** command.

Annotate Views

Annotate Views lets you automatically annotate manually created drafting views based on the KF rules and automation preferences embedded in the drawing template used to create the view.

You can delete and recreate all existing annotation in the views, or add annotations to newly created views while preserving the automated annotations originally placed on the drawing.

Note To use this tool, the template used to create the drawing must have automation rules applied to it via the **Tools→Drawing Format→Automation Rules** command.

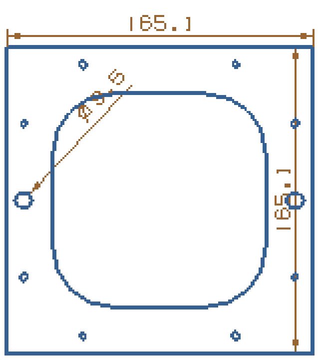
Where do I find it?

Application	Drafting
Menu	**Tools→Drawing Automation→Annotate Views**

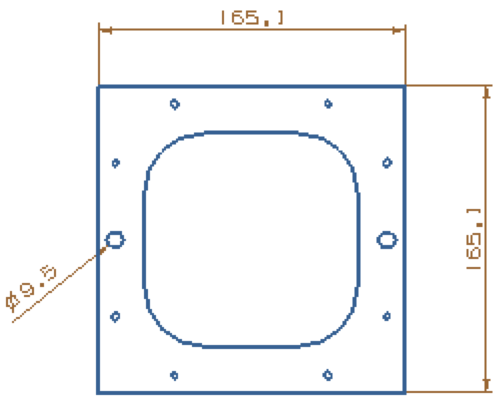
Distribute Annotations

Distribute Annotations places dimensions in relative locations to each other and to other objects in a view. This command is intended primarily for annotations automatically generated from a drawing template. You can also use it for dimensions created with the **Annotate Views** command as well as for those created manually.

When you distribute annotations in a view, NX applies the values and rules on the **Annotation Distribution Rules** tab of the **Drafting Automation Preferences** dialog box to the dimensions in that view.



Before Distribute Annotations applied



After Distribute Annotations applied

Where do I find it?

Application	Drafting
Menu	**Tools→Drawing Automation→Distribute Annotation**

Import AutoCAD Block

Use **Import AutoCAD Block** to create custom symbols by importing block definitions from AutoCAD DXF and DWG files.

There are two methods for selecting AutoCAD files with blocks:

- Select an AutoCAD file and its list of blocks will be extracted.
- Select a folder and any AutoCAD files open and any list of blocks is extracted.

Users can mix any selection of files and folders, the blocks tree list will accumulate all selections.

By default the block name will be used to define the symbol name, and the symbol part file name will be the symbol name with a .sym.prt extension (not applicable to part symbols).

The **Import Summary** will show the number of blocks processed, number of symbols created and their status.

To remove a new symbol, you can delete it after import like any other custom symbol.

You can recreate symbols for simple blocks as you migrate AutoCAD data to NX Drafting. However, in the case of more complex blocks or large block libraries, it is easier to use this functionality to create the NX symbol from the existing AutoCAD blocks. In the event of symbol name clashes, a number will be appended to the name.

Note See DXF/DWG import interface for more information about importing AutoCAD data.

Where do I find it?

Application	Drafting
Prerequisite	The DXF / DWG Translator (dxfdwg kit) must be installed.
Menu	**File→Import→AutoCAD Block**
Resource bar	**Reuse Library**  tab ® **Custom Symbol Library** ® right-click a folder ® **Import AutoCAD Block**

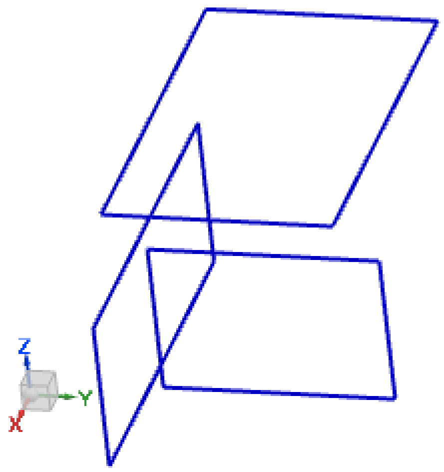
Copy to 3D enhancements

What is it?

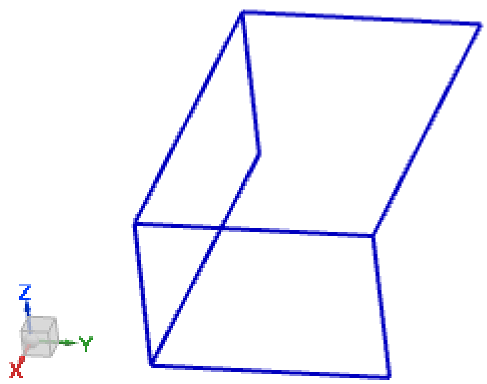
Several enhancements have been made to the **Copy to 3D** command.

- You can now output selected curves as sketches, basic curves, sketches and solid body, or component parts from groups.
- When you specify an automatic placement location, you can use the **Automatically Reposition Geometry** option to have NX automatically reposition the geometry for better alignment.

Previously, when geometry in drawing views was properly dimensioned but not aligned, the output was not aligned in Modeling.



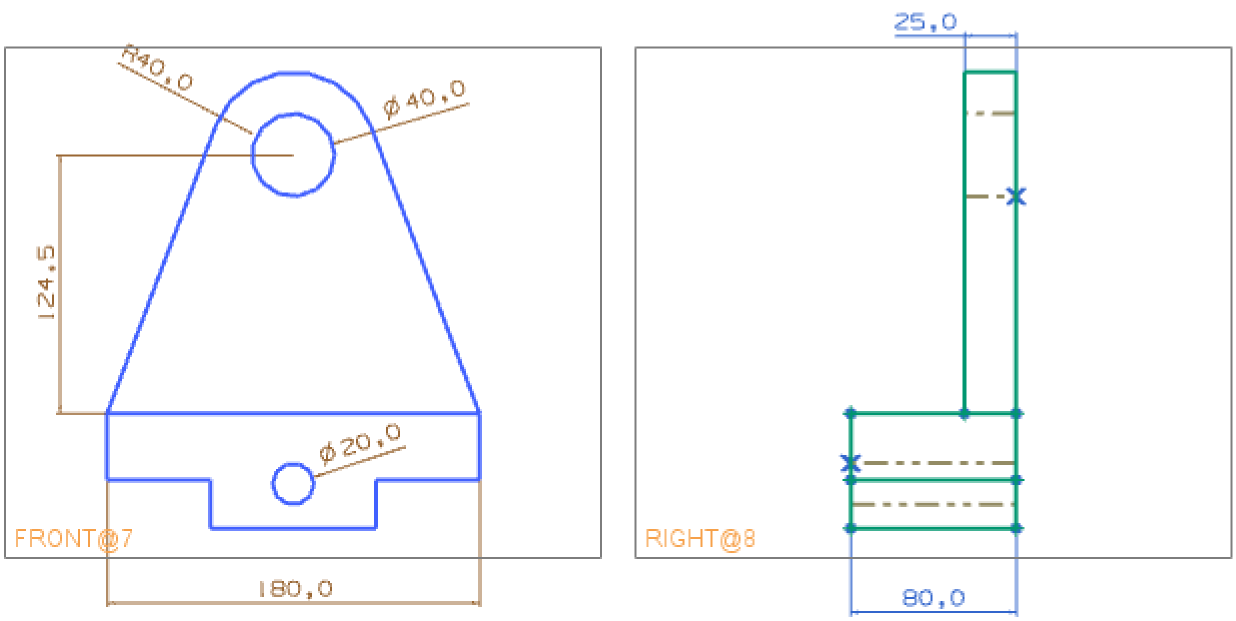
The **Automatically Reposition Geometry** option allows you to automatically place geometry in Modeling.



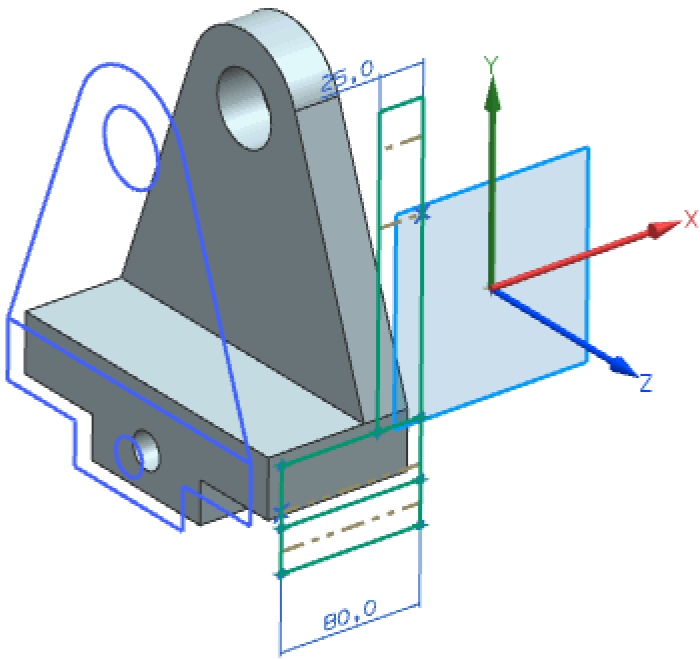
Output

When **Output** is set to **Sketches**, **Sketches and Solid**, or **Component Parts from Groups**, modeling sketch dimensions will be generated corresponding to the input drafting sketch dimensions.

In the following graphics the output is set to **Sketches and Solid**.



Before Copy to 3D



After Copy to 3D

Where do I find it?

Application	Drafting
Menu	**Tools** ® **Copy to 3D**
Shortcut Menu	Right-click selected Drawing views ® **Copy to 3D**
Part Navigator	Right-click one or more Drawing view nodes ® **Copy to 3D**

Custom symbol enhancements

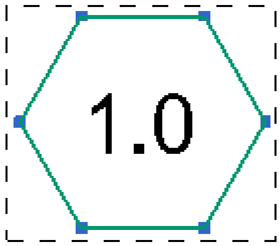
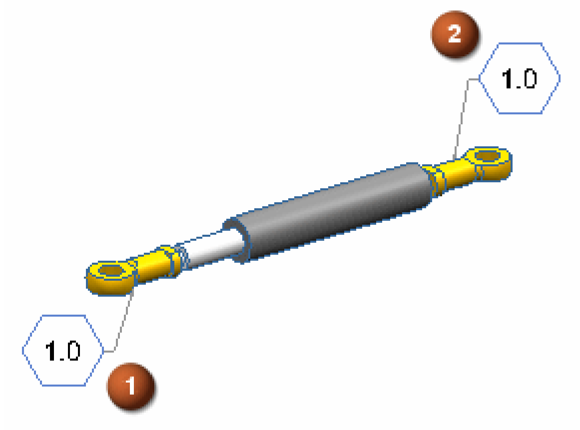
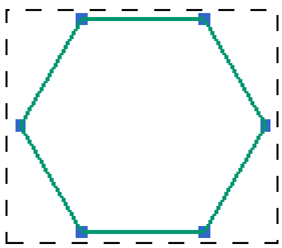
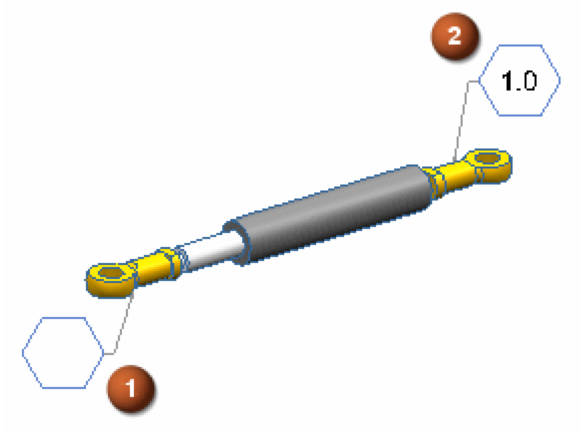
Associative custom symbols

When you create custom symbols from a master custom symbol definition, you can make the custom symbols associative to the master custom symbol. If the master custom symbol changes, the associative copies of the symbol also change.

Note Associativity is not supported for nested custom symbols or PMI custom symbols inherited onto a drawing.

When you create associative custom symbols, it is important to note the following.

- Associativity to the master symbol is lost if the master symbol is deleted, the folder containing the master symbol is deleted, or the symbol instance is smashed into its constituent parts. You cannot add master symbol associativity to existing symbol instances created in previous NX releases.
- When you create an instance of a master custom symbol, the **Lock Update** option is available. When set, the symbol instance is locked, and changes to the master symbol definition do not affect the instance.

 Original master symbol	 Symbol instances from original master symbol
 Changed master symbol	 Symbol instances from changed master symbol

1 – Unlocked symbol 2 – Locked symbol

- You can manually update symbol instances that are unlocked.

  See **Out-of-Date** Folder for additional information.

- You can also change the locked status of existing custom symbol instances.

- The **Reassociate** command lets you reassociate a symbol instance to its master symbol definition. You can only use the **Reassociate** command on symbol instances created in NX 8 and above.

Create Associative Symbols customer default and the Lock Update option

A new customer default, **Create Associative Symbols**, determines whether or not to create an associative custom symbol instance. It also determines the visibility and availability of the **Lock Update** option in the **Custom**

Symbol dialog box. When **Create Associative Symbols** is selected, the **Lock Update** option is visible and available, and you can create both associative and non-associative symbol instances. When it is not selected, the **Lock Update** option is not visible and symbol instances are always created as non-associative copies of the master custom symbol.

Why should I use it?

Create associative custom symbol instances when you want your symbol instances to always reflect the master definition of the original custom symbol.

Where do I find it?

Lock Update option

Application	Drafting and PMI
Toolbar	In the Drafting application: **Symbol**® **Symbol Drop-down** list® **Custom Symbol** In the PMI application: **PMI**® **PMI Symbols Drop-down**® **Custom Symbol**
Menu	In the Drafting application: **Insert**® **Symbol**® **Custom** In the PMI application: **PMI**® **Symbol**® **Custom**
Location in dialog box	**Settings** group® **Lock Update**

Create Associative Symbols customer default

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Drafting**® **Custom Symbols**® **All** tab® **Create Associative Symbols**

Reassociate command

Application	Drafting and PMI
Graphics Window	Right-click custom symbol instance® **Reassociate**

Edit Custom Symbol

You can edit the definition of a master custom symbol by using the **Edit Custom Symbol** command. You can also edit the master custom symbol by directly opening the master symbol part, or in the **Reuse Library** by right-clicking the symbol and choosing **Edit**.

- You can only edit a master custom symbol in the Drafting environment.

  Note If the symbol was created as a 3D object, NX will attempt to display the symbol as a 2D object in the Drafting environment. If it cannot be displayed, a warning is displayed and you cannot edit the symbol.

- You can change all constituents of the master custom symbol except the symbol name and the symbol location.

- You can only edit those symbols for which you have write permission.

- You must use the **Finish Custom Symbol Edit** command to save any geometric and annotation changes to the master symbol definition.

- You can use the **Exit Custom Symbol Edit** command to exit the custom symbol editing environment without saving your changes.

- After clicking **Finish Custom Symbol Edit**, you can change the symbols anchor points, leader line points, symbol icon image, and any text attributes in the **Finish Symbol Edits** dialog box. When you click **OK** in the **Finish Symbol Edits** dialog box all edits are saved and the master symbol part is closed.

Use the **Custom Symbols** node in the **Out-of-Date** folder in the **Part Navigator** to locate and manually update unlocked instances of the master custom symbol in your parts.

Where do I find it?

Application	Drafting
Prerequisite	You must be able to edit your master custom symbol in the Drafting environment
Toolbar	**Symbol**® **Symbol Drop-down** list® **Edit Custom Symbol**
Menu	**Edit**® **Symbol**® **Edit Custom Symbol**
Resource bar	Reuse Library tab® **Member Select** panel® right-click a custom symbol® **Edit**

Finish Custom Symbol Edit

Use **Finish Custom Symbol Edit** to do the following.

- Save all geometric and annotation changes you made to a master custom symbol.
- Create a new snapshot of the master custom symbol.
- Change the leader attachment type and attachment point.
- Re-specify any text parameters.

The **Finish Custom Symbol Edit** command is only available when you are editing the definition of a master custom symbol.

See **Edit Custom Symbol** for more information.

Where do I find it?

Application	Drafting
Toolbar	**Symbol**® **Symbol Drop-down** list® **Finish Custom Symbol**
Menu	**Edit**® **Symbol**® **Finish Custom Symbol Edit**

Exit Custom Symbol Edit

Use **Exit Custom Symbol Edit** to exit the custom symbol editing environment without saving your geometric or annotation changes.

The **Exit Custom Symbol Edit** command is only available when you are editing the definition of a master custom symbol.

See **Edit Custom Symbol** for more information.

Where do I find it?

Application	Drafting
Toolbar	**Symbol**® **Symbol Drop-down** list® **Exit Custom Symbol**
Menu	**Edit**® **Symbol**® **Exit Custom Symbol Edit**

Define Symbol from Catalog

What is it?

Use the **Define Symbol from Catalog** command to create a 2D custom symbol in the **Custom Symbol Library** from the **Fasteners** catalog. Optionally, you can immediately add the created symbol instance to your drawing using the **Custom Symbol** dialog box.

While defining the symbol from the catalog item, you can do the following:

- Browse the catalog and select the catalog item to create the 2D custom symbol.
- Set individual parameters for the item being created.
- Choose the view orientation in which the 2D custom symbol is created.
- Choose to create centerlines for the 2D custom symbol.
- Identify in what custom symbol library folder you want to place the symbol once it is created.
- Name the created 2D custom symbol or use automatically generated name.

The **Fasteners** catalog contains fastener items that are classified by standard and by type. The size and length characteristics of items are stored parametrically in the catalog. The items are not stored in symbol part files, but their information is stored in the catalog XML file.

Only 2D custom symbols can be created.

Why should I use it?

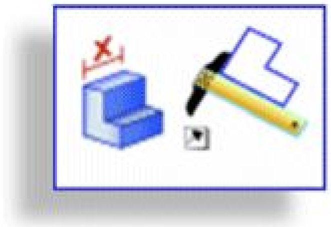
Use the **Define Symbol from Catalog** command to create only the 2D custom symbols you need from the catalog items. This saves disk space.

Where do I find it?

Application	Drafting
Toolbar	**Symbol® Define Symbol from Catalog**
Menu	**Insert® Symbol® Define Symbol from Catalog**

Drafting welcome

After you install NX, a welcome page automatically opens from the Resource bar the first five times you log in and enter Drafting. This welcome page provides general information, tips, and tutorials for using the Drafting environment.



Recommended Setup

Under **Recommended Setup**, **Specify my Drafting Standard** allows you to preset all of the drafting preferences in your session according to one of the NX-supplied Drafting standards or to a customized standard.

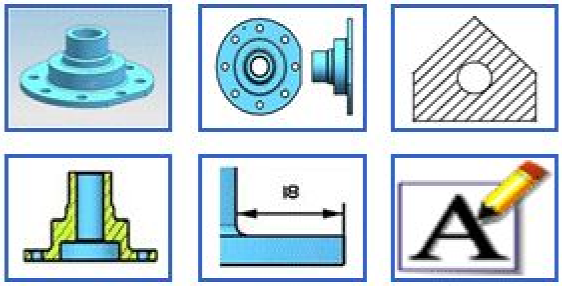
Overview of NX Drafting

Under **Overview of NX Drafting**, you can review the primary components of the graphic user interface in Drafting, or open the What's New Guide for more information on the latest release of NX.

Tutorials

If you are a new user, you can familiarize yourself with the fundamentals of Drafting by viewing online tutorials that cover:

- Creating new drawing parts and sheets.
- Adding base and orthographic views of a 3D model.
- Creating section views, detail views, and view breaks.
- Adding and editing dimensions.
- Creating and editing annotations and centerlines.
- Making a standalone drawing and sketching in Drafting.



Each tutorial is supplied with a sample part so that you can complete the activity.

Resetting the welcome page

After the fifth login, you can restore the welcome page to the Resource bar by clicking **Reset Welcome Page** on the **General** page of the **Drafting Preferences** dialog box.

Customer defaults

You can use two customer defaults to control the appearance and behavior of the welcome page:

Drafting Welcome Page Duration

Sets the number of log-ins for which the welcome page appears.

Display Recommended Setup in Welcome Page

Displays the Recommended Setup section, under which the **Specify my Drafting Standard** option appears.

See the **Miscellaneous** tab under **Drafting→General**.

Where do I find it?

Application	Drafting
Resource bar	**Internet Explorer**  tab

Drafting user interface improvements

What is it?

The user interface for the Drafting application is simplified to improve usability and make the system more intuitive. Enhancements include:

- Non-drafting commands on menus and toolbars are removed. For example, some commands are only visible when the sheet display is turned off.

- The drafting commands have been reorganized so that they are easier to find.
- The toolbar and menu icons are consistent between Drafting and PMI.
- The default background color is white and most drafting object colors are black.
- Out-of-the-box drawing templates are updated and made more intuitive. There will be a single set of drawings for each template type, regardless of the relationship type.
- The base view part defaults to the loaded, or partially loaded, master model part. If the master model is not loaded, the base view part defaults to the current drawing part and an alert is issued. The options in the list **Model View to Use** are derived from the part specified in the **Part** group.
- A new relationship filter on the **File**→**New** dialog box makes creating a 2D type drawing intuitive.

The Relationship filter

In Native Mode, in the **New** dialog box, on the **Drawing** page, there is a new **Relationship** filter, which sorts the list of available drawing templates.

The following options are available on the **Relationship** filter:

- **Reference Existing Part** – The template list shows only those templates that support referencing existing parts. For example, you can choose this option to create a drawing of an existing part or a manufacturing operation on an existing part.
- **Stand-alone Part** – The template list shows only those templates that support the creation of a new stand-alone part. For example, you can choose this option to create a new model, or to create a drawing that does not reference an existing model.
- **All** – The template list shows all the templates.

Why should I use it?

The enhancements make it easier to use NX. You or your system administrator can configure the user interface enhancements to match your company's requirements and preferences.

Where do I find it?

New dialog box

Application	Drafting
Toolbar	**Standard→New**
Menu	**File→New**

Support for Standard Fonts

Standard font types available in the FreeType Font library are supported in the PMI and Drafting environments.

Note Although the FreeType Font library supports non scalable and other font types, only scalable TrueType, OpenType, and PostScript fonts are supported.

When setting your character font, it is important to remember the following points:

- Existing NX font types are still available and are located in a directory set by the **UGII_CHARACTER_FONT_DIR** environment variable.
- Standard Fonts files are found in your normal font directories, usually in the *C:\Windows\Fonts* on Windows or in a configuration file on Unix.
- You can use custom fonts by placing the font files in a directory referenced by the **UGII_STANDARD_FONT_DIR** environment variable.
- NX will search for the character font in the following order:
  1. Standard font location (that is, *C:/Window/Fonts*).
  2. The directory specified by **UGII_STANDARD_FONT_DIR**.
  3. The directory specified by **UGII_CHARACTER_FONT_DIR**.
- A default font is used when a font cannot be found. The default font can be changed, but is initially set to Arial Unicode MS. If Arial Unicode MS is not found, the default font is set to Tahoma. If Tahoma is not found, then the default font is set to Arial.

Why should I use it?

Use Standard Fonts to replace, enhance, or supplement your current set of drafting and PMI character fonts.

Where do I find it?

All fonts found in standard directories and in the custom directory are automatically available in customer defaults, preference dialog boxes, style dialog boxes, and any other dialog box that lets you specify a character font.

Cut, copy, and paste enhancements

What is it?

The cut, copy, and paste capabilities in Drafting have been enhanced. You can now copy and paste the following Drafting objects:

- Sheets and drafting views between parts.
- Associative and retained annotations.
- PMI-inherited annotations, view labels, and section line letters.

You can also:

- Reassociate an object when you use **Paste**.
- Trim pasted sketch curves using the boundary of the original drawing member view.
- Copy view-dependent edits of sketch curves.

Note Copies of associative or retained annotations will always be on the current sheet, though they may be associated to other copied geometry in the destination member view.

Completely unassociated annotations, and dimensions which are not retained, which were copied in NX 7.5 will still be pasted into the destination view or sheet.

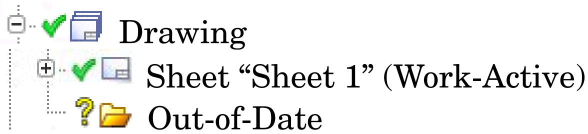
Where do I find it?

Application	Drafting
Toolbar	**Standard→Cut** or **Copy** or **Paste**
Menu	**Edit→Cut** or **Copy** or **Paste**
Graphics window	Right-click on a drawing member view ® **Cut** or **Copy** or **Paste** Right-click on a drawing sheet® **Copy** or **Paste**

Out-of-Date folder

What is it?

In the **Part Navigator**, an **Out-of-Date** folder is available under the **Drawing** node.



This folder serves as a container for following types of drafting objects:

- Custom symbols
- Parts lists
- Tabular notes

Out of date checks

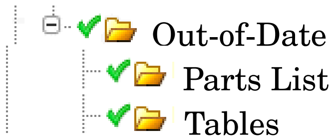
The empty **Out-of-Date** folder initially appears with a question mark (?) symbol in front of it.

When you place a supported annotation type on the drawing, a sub-folder for the annotation is added under the **Out-of-Date** folder. The question mark indicates that an out of date check has not yet been performed on the annotation.

As objects become out of date, they are marked in the navigator with the stop watch symbol.

Updating Objects

You can use the **Out-of-Date** folder to update the objects listed in it. When an object is up to date, it is removed from its respective folder, and a green check mark symbol appears on that node.



Shortcut menu commands

You can use shortcut menu commands on the objects in the **Out-of-Date** folder to do the following:

Update All	Updates all the out of date objects under the selected folder.
Refresh	Expands the Out-of-Date folder when you return to the Drafting application. You can choose Refresh again to perform an out of date check on any of the folders. Also removes any unneeded folders from the Out-of-Date folder.
Refresh All	Performs an out of date check on the contents of the selected folder.
Collapse All	Collapses the contents of the selected folder.
Expand All	Expands the contents of the selected folder.

Why should I use it?

Use the **Out-of-Date** folder to identify and update objects whose out of date status may not be readily apparent on the drawing sheet. This is especially useful in cases where automatic updates have been turned off for parts lists and tabular notes. Likewise in the case of custom symbols, NX does not automatically check to see if a symbol is out of date with respect to its master symbol geometry. You can use the **Out-of-Date** folder to manually check for symbol updates.

Where do I find it?

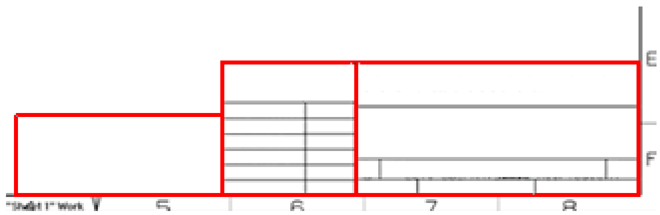
You can access the **Out-of-Date** folder from the **Part Navigator**.

Title Block

Use the **Title Block** command to construct a customized title block that you can use in your drawing template or part file.

You can:

- Create a single title block entity comprised of one or more tabular notes, pre-arranged in the positions you want on the drawing sheet. The cells of the tabular note can have a designated annotation type or mixture of annotation types. Annotation types are limited to simple text, symbols, images, or associative text such as expressions or attributes.



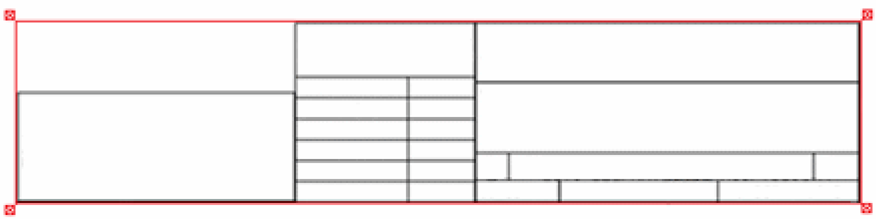
Title block comprised of three tabular notes

- Designate a cell as locked or unlocked. When the title block is imported as part of a template, the contents of a locked cell cannot be edited.

- Provide labels for each cell in the title block, which are used to identify cells during the edit process.

- Designate an alignment position for moving and locating the title block.

After you select all the tables for your title block, a bounding box is calculated around the area of the title block. The table's alignment position is derived from one of the four corners of the bounding box.



Note

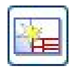
- Non-tabular notes (such as parts lists, hole tables, and routing tables), tables with more than one section, tables which are in drafting views or modeling views, and tables with leaders cannot be included in a custom title block. Also, the **Auto Size Row**, **Auto Size Column**, and **Wrap** fit methods are not supported. **Auto Size Text** is used instead.

- Because you can enter only one piece of information per cell, any cell in a title block containing more than one attribute, expression, or other object is automatically locked.

- After the title block is defined and imported as part of a template into another drawing, you can set the default to prompt you for manual input to the title block.

- You should consider on which layer to create the title block. You might consider putting the title block on a layer that is normally not selectable. For example, with the NX supplied templates, the drawing borders and title blocks are always found on layer 256.

Where do I find it?

Application	Drafting
Toolbar	**Drawing Format→Title Block** 
Menu	**Tools→Drawing Format→Title Block**

Populate Title Block

Use the **Populate Title Block** command to edit the contents of individual unlocked title block cells.

You can:

- Use the **Populate Title Block**  command to add content to the title block In a template part,

- Right-click the title block and use the **Populate** command to add content to the title block In an instantiated template.

Editing title block contents when instantiating a drawing template

To edit the content of a title block while instantiating a drawing template, you must set the **Display Populate Title Block Dialog on Template Instantiation** option in the **Customer Default** dialog box. This default is on the **Title Block** tab under **Drafting→Drawing**

The **Populate Title Block** dialog box opens when you instantiate a drawing template. You can then edit the contents of any unlocked cell.

Where do I find it?

Application	Drafting
Toolbar	**Drawing Format® Populate Title Block**
Menu	**Tools→Drawing Format→Populate Title Block**
Shortcut Menu	Right-click the title block® **Populate**

Editing title blocks

Editing title block definitions

To edit the label or lock status of a cell:

- Right-click the title block and choose **Edit Definition**.

The **Edit Definition** command opens the **Define Title Block** dialog box.

To re-size rows or columns, or to merge cells, you must first remove the table from the title block.

After you make your edits, add the table back to the title block template by clicking **Select Tables**, and then selecting the table.

Removing a table from the title block

1. In the **Define Title Block** dialog box, click **Select Tables**.

2. Hold Shift and select the table.

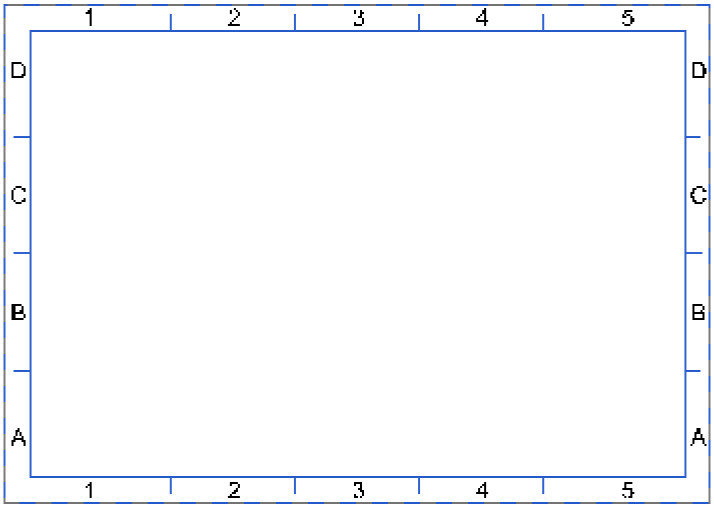
You can remove all but the last remaining table from the title block. If there is only one table in the title block, you first need to add a temporary table to the title block, and then remove the table you wish to edit. When you make the needed changes to the table, add it back to the title block. Afterward, remove the temporary table.

Where do I find it?

Application	Drafting
Shortcut menu	Right-click the title block® **Edit Definition**

Borders and Zones

The **Borders and Zones** command adds associative borders and zones to the active drawing sheet in your part. The border of a drawing is the line that defines the outer boundary of the drawing sheet. Drawing zones are separate, rectangular cells on the drawing sheet which are lettered in the vertical direction and numbered in the horizontal direction.



If you mark the drawing as a template, the associative border and zone data is saved with it.

Note You cannot edit borders or zones in legacy parts. You must first delete the existing border and zone information, and then create new borders and zones with this command. If you import border and sheet zones from a template part, you cannot edit them.

The color, font, and width settings for border lines and zone marking lines are based on the customer defaults and preferences for general NX curves. The color, font, and width settings for zone labels are based on the customer defaults and preferences for general text annotation.

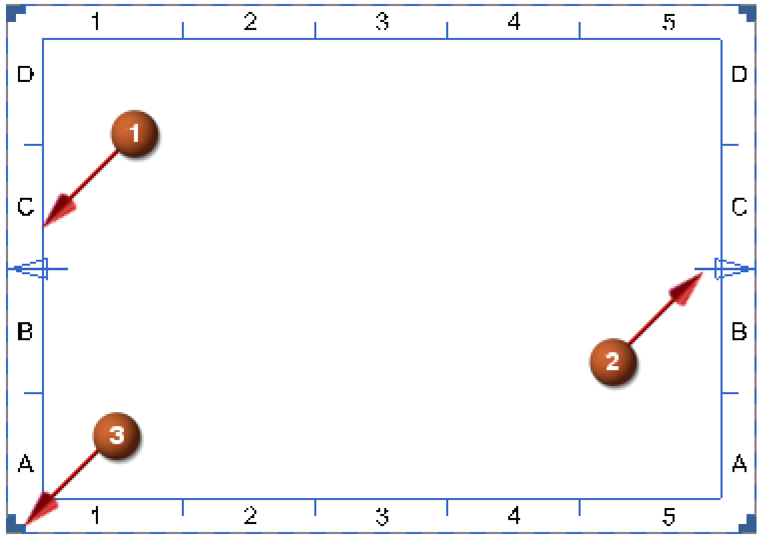
Borders

For borders you can:

- Control their appearance and width (1).

- Control the appearance and extension size of horizontal and vertical centering marks (2).

- Control the appearance and size of trimming marks (3).



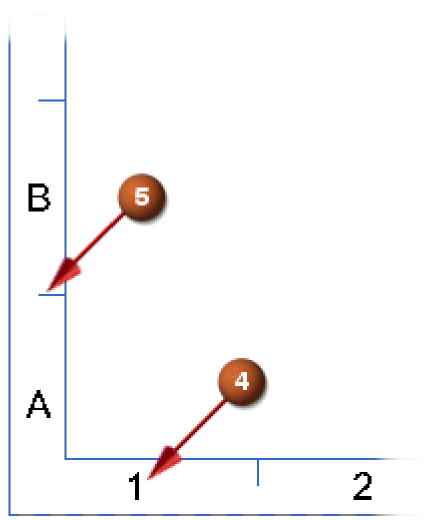
Zones

You can set the zone type to:

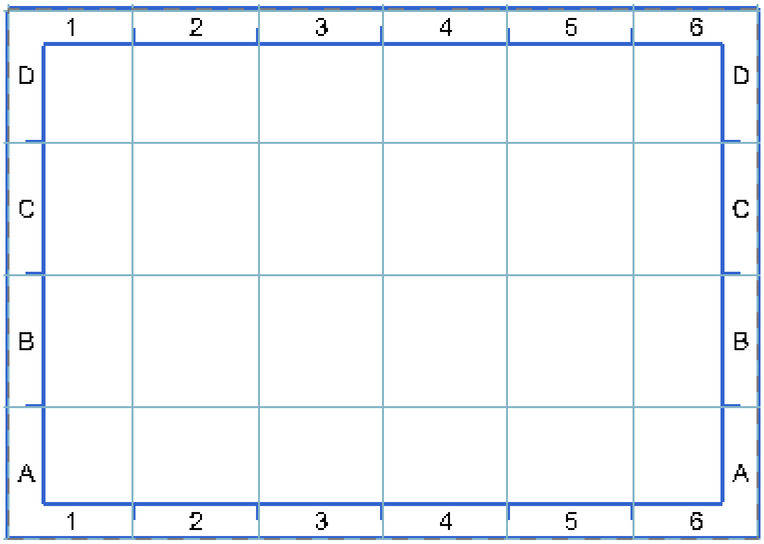
None	No drawing sheet zones are created.
Standard	Drawing sheet zones are created according to the current Drafting Standard. This is the default.
Custom	Lets you specify unique zone parameters such as size, origin, and margin.

For **Standard** and **Custom** zones, you can:

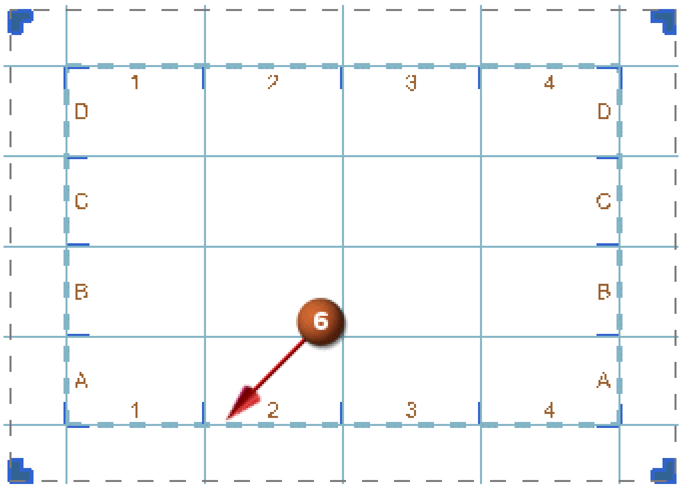
- Control the appearance, size, and font of zone labels (4).
- Control the appearance and size of zone dividing lines (5).



- Display the zone grid when you set the **Use Sheet Zone Grid**.

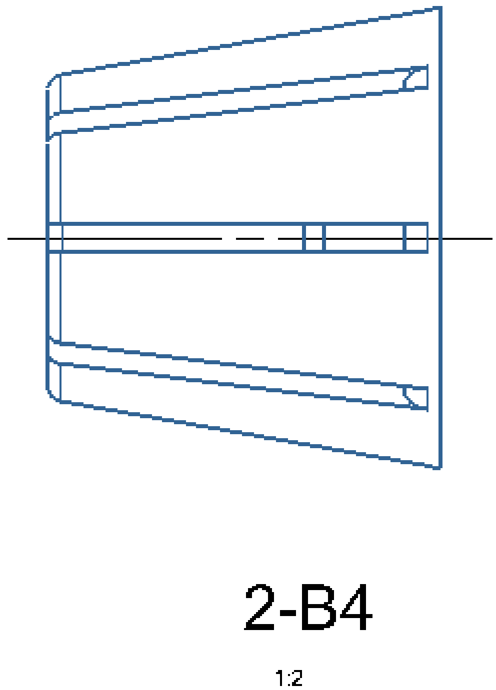


- Control the appearance and size of margins which limit zones to a specific portion of the drawing sheet. If margins are specified and the sheet zone grid is on, a dotted box representing the sheet zone area is displayed (6).



- Display the associative sheet zone reference in view labels and section line symbols.

  Note The **View Label** option in the **View Label Preferences** dialog box must be set in order to display associative sheet zone reference information in views and section line symbols.



- Use the **Insert Sheet Zone** option to insert an associative sheet zone reference of a view into a note, table, or symbol. For example:

  1. Refer to 2B4,

  where 2B4 is the sheet number and zone reference of a specific view.

  The **Insert Sheet Zone** option is located in the any annotation command that supports the use of the **Text Editor**  for textual content.

Note You should consider on which layer to create the borders and zones. You might consider putting the them on a layer that is normally not selectable. For example, with the NX supplied templates, the drawing borders and title blocks are always found on layers 255 and 256.

Where do I find it?

Application	Drafting
Toolbar	**Drawing Format→Borders and Zones**
Menu	**Tools→Drawing Format→Borders and Zones**

Drawing sheet numbers and revisions

What is it?

You can assign a name, number and revision to individual sheets in a drawing. The sheet number may contain a primary index, a delimiter, and a secondary index such as 1-A.

Sheet numbers and revisions appear on the **Insert Sheet** and **Edit Sheet** dialog boxes where you can modify their values. The sheets are displayed in the **Part Navigator** in sheet number order. Revision letters also appear in the **Part Navigator** where they can be incremented.

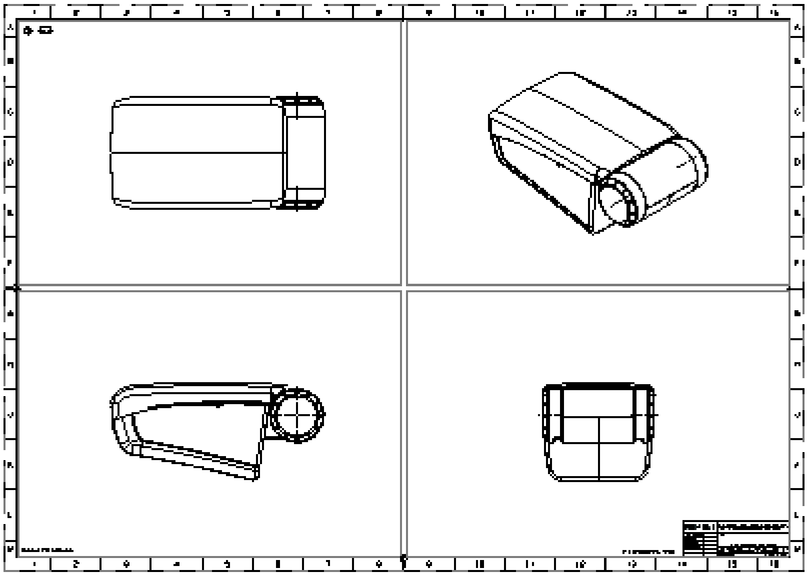
Customer defaults for automated sheet numbering appear in the **Customer Defaults** dialog box, under **Drafting→General**, on the **Drawing** tab. These settings also appear on the **Sheet** tab of the **Drafting Preferences** dialog box.

Where do I find it?

Application	Drafting
Toolbar	**Drawing→New Sheet** **Drawing Edit→Edit Sheet**
Menu	**Insert→Sheet** **Edit→Sheet**
Location in dialog box	**Name** group® **Sheet Number** **Name** group® **Revision**

View Creation Wizard

The **View Creation Wizard** simplifies the process of adding one or more drafting views to a drawing sheet.



The wizard steps you through the following process of:

- Selecting a currently loaded, recently loaded, or unloaded part or assembly.
- Specifying the preview style and view display parameters.
- Optionally inheriting existing PMI data from the assembly model.
- Optionally selecting existing assembly arrangements.
- Specifying a view orientation for the parent view.
- Constructing a multi-view layout.

Note The **View Creation Wizard** is available for master model drawings only. It is not available for standalone drawings.

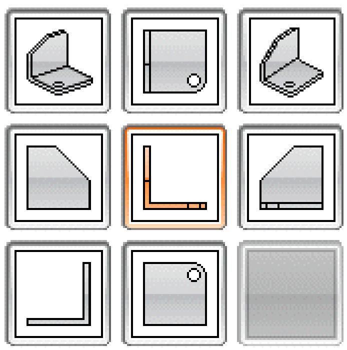
By default, the **View Creation Wizard** is launched immediately after creating a new drawing or a new drawing sheet. You can change this default behavior by adjusting the options on the **General** tab of the **Drafting Preferences** dialog box, or by adjusting the options on the **Workflow** tab under **Drafting→Drawing** on the **Customer Defaults** dialog box.

The View Creation Wizard window

The options in the **View Creation Wizard** window step you through the process of defining and placing a group of views on the current drawing sheet.

You can:

- Select the part or assembly to be depicted on the drawing.
- Set your preferences for the display. You can click **View Style** to set view style preferences not directly available from the from the **View Creation Wizard** window.
- Specify the parent view orientation.
- Configure a view layout. You can include up to six standard orthographic views in the layout, along with an isometric or trimetric view. The projection angle of the views is derived from the projection angle of the drawing.



- Place the view layout on the drawing sheet using **Margins** to define the minimum distance between adjacent view borders in the layout, and between view borders and the edge of the drawing sheet.

If the **View Placement** option is set to **Automatic**, you can complete the placement of the views in the first step. Use the **Manual** option if you want to place the center of the view layout anywhere on the drawing sheet.

Where do I find it?

Application	Drafting
Toolbar	**Drawing→Add View Drop-down** list® **View Creation Wizard** **Drawing→View Creation Wizard**
Menu	**Insert→View→View Creation Wizard**

Sheet dialog box:

Location in dialog box	**Settings** group® **Automatically Start View Creation**® **View Creation Wizard**

View Creation Wizard preferences:

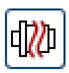
Menu	**Preferences→Drafting**
Location in dialog box	**General** tab® **Model–based Drawing Workflow**® **View Creation Wizard**

View Creation Wizard customer defaults:

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Drafting**® **General**® **Drawing** tab® **Model–based Drawing Workflow**® **View Creation Wizard**

View Break

What is it?

You can use the **View Break**  command to add multiple horizontal or vertical breaks to a view. Two types of view breaks are available:

Regular

The view break has two break lines representing the conceptual gap on the drawing.

Single-Sided

The view break has only one break line, and only one side of the break is visible.

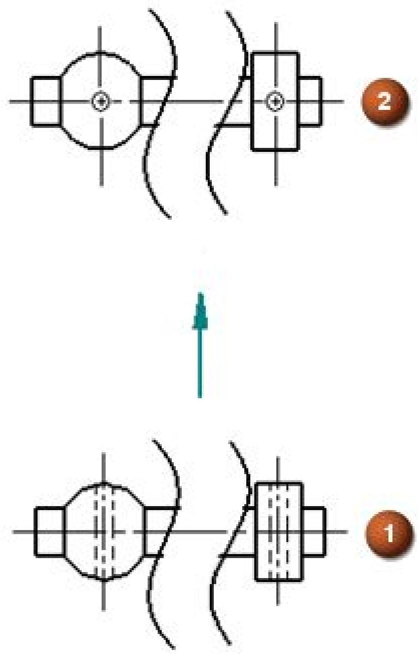
For the first view break in a view, the direction of the break lines can be horizontal, vertical, or general. NX sets the default break direction to horizontal if the geometry is wider than it is high on the sheet, otherwise it sets it to vertical. You can also specify a vector to set a general direction for the break line. When you add a new break to a view that already has a view break, the direction must be either parallel or perpendicular to the previous breaks.

You can add breaks to the following views:

- Base views.
- Projected views.
- 2D drawing views.
- Section views with simple or stepped section line symbols.

Note You cannot add breaks to detail views, perspective views, and legacy broken views using the **View Break** command.

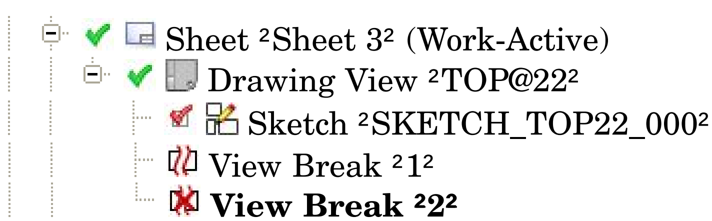
When the parent view (1) already has a view break, you can create projected views (2) and section views with view breaks that NX creates automatically.



The view breaks created in the projected and section views are independent and can be modified and deleted.

Note The **Propagate View Break** customer default controls whether or not the view break propagates to projected and section views.

The breaks associated to a view are displayed in the **Part Navigator**.



You must suppress the view break to modify the sketch or the boundary of the view.

Why should I use it?

View breaks allow more compact views to be drawn on the drawing sheet, so that long areas of the geometry can be omitted and areas of interest can be shown and documented. The new **View Break** command allows streamlined creation of view breaks.

Where do I find it?

Application	Drafting
Toolbar	**Drawing**→ **Add View Drop-down** list® **View Break** **Drawing**→ **View Break**
Menu	**Insert**→**View**→**View Break**
Graphics window	Right-click the view border® **Add View Break**

Set **View Break** defaults

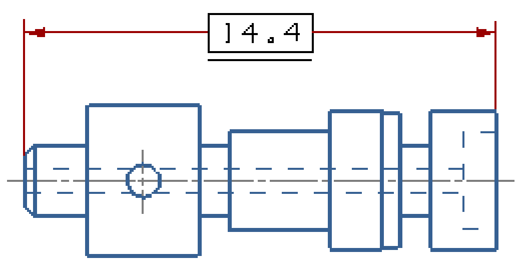
Customer Defaults dialog box	**File**→**Utilities**→**Customer Defaults**→**Drafting**→**General**→**View Break** tab

ASME Y14.5-2009 Drafting Standard enhancements

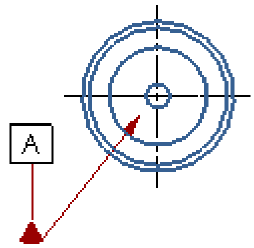
What is it?

New options are available to create ASME Y14.5-2009 compliant annotations. Enhancements to the Drafting and PMI annotation functionality let you do the following:

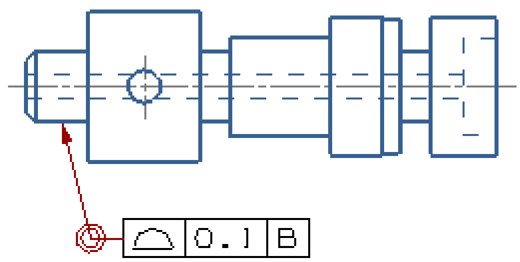
- Create basic dimension types that are not to scale.



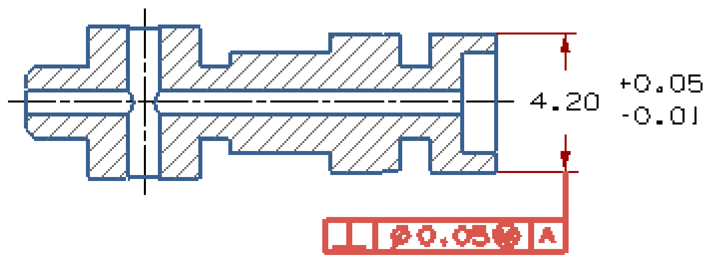
- Create a **Datum on Dot Terminated** leader with an arrowhead for Feature Control Frames (FCS), Datum Feature Symbols (DFS), and notes.



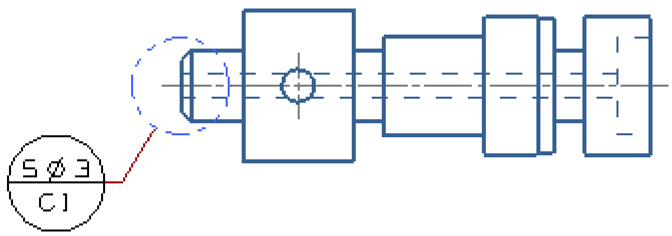
- Create a leader with an all over symbol for a feature control frame.



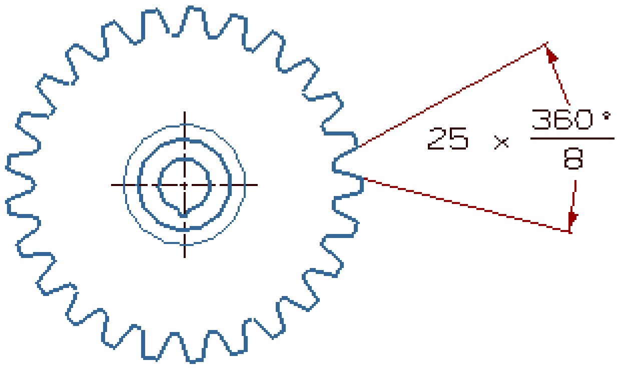
- Correctly attach annotations, such as a feature control frame, to a dimension using a **Flag** leader.



- Create a datum target symbol with a spherical zone shape.



- Specify an angular dimension as a fractional division of an arc.



Where do I find it?

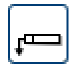
Basic, Not to Scale dimension type

Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation® Annotation Preferences**
Menu	In the Drafting and PMI applications: **Preferences® Annotation**
Location in dialog box	**Dimensions** tab® Dimension Display Type list® **Basic, Not to Scale**

Datum on Dot Terminated leader with an arrowhead

Application	Drafting
Toolbar	**Annotation→Datum Feature Symbol** **Annotation→Feature Control Frame** **Annotation→Note**
Menu	**Insert® Annotation® Datum Feature Symbol** **Insert® Annotation® Feature Control Frame** **Insert® Annotation® Note**
Location in dialog box	**Leader** group® **Type® Dot Terminated** **Style** subgroup® **Arrowhead**

Leader with an **All Over** symbol for a feature control frame

Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation® Feature Control Frame**  In the PMI application: **PMI® PMI Annotation Drop-down® Feature Control Frame** 
Menu	In the Drafting application: **Insert® Annotation® Feature Control Frame** In the PMI application: **PMI® Feature Control Frame**
Location in dialog box	**Leader** group® **Type**® **All Over**

Datum target symbol with a spherical zone shape

Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation® Datum Target**  In the PMI application: **PMI® Datum Target** 
Menu	In the Drafting application: **Insert® Annotation® Datum Target** In the PMI application: **PMI® Datum Target**
Location in dialog box	**Type** group® **Spherical**

Angular dimension as a fractional division of an arc

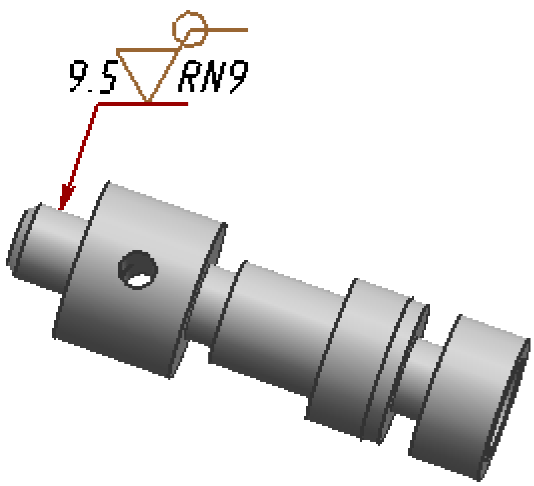
Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation**® **Annotation Preferences**
Menu	In the Drafting and PMI applications: **Preferences**® **Annotation**
Location in dialog box	**Units** tab® **Display as a Fraction**

ESKD Drafting Standard enhancements

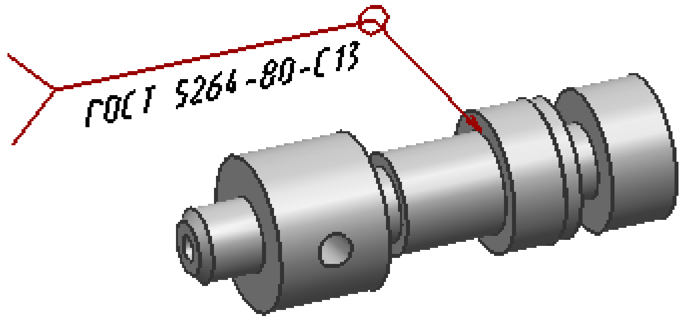
The following enhancements are provided in both Drafting and PMI to better support the ESKD drafting standard.

Note The ESKD standard must be set in customer defaults, or in **Surface Finish** option on the **Symbols** tab of the **Annotation Preferences** dialog box.

- You can specify surface finish symbols with roughness values in which the lay symbol and roughness are in a single line.



- The **Not Specified** weld type lets you create a symbol with just weld properties and no specific weld type.



Where do I find it?

Surface finish symbol with a single line roughness callout

Application	Drafting and PMI
Prerequisite	Your drafting standard must be set to **ESKD** in the **Annotation Preferences** dialog box.
Toolbar	In the Drafting application: **Annotation® Surface Finish Symbol** In the PMI application: **PMI® Surface Finish**
Menu	In the Drafting application: **Insert® Annotation® Surface Finish Symbol** In the PMI application: **PMI® Surface Finish® Attributes** group® **Standard® ESKD**
Location in dialog box	**Attributes** group® **Single Roughness Value**

Not Specified weld type

Application	Drafting and PMI
Prerequisite	Your drafting standard must be set to **ESKD** in the **Annotation Preferences** dialog box.
Toolbar	In the Drafting application: **Annotation® Weld Symbol**  In the PMI application: **PMI® Weld Symbol** 
Menu	In the Drafting application: **Insert® Annotation® Weld Symbol** In the PMI application: **PMI® Weld Symbol**
Location in dialog box	**Other Side** group® Weld Symbol list® **Not Specified** **Arrow Side** group® Weld Symbol list® **Not Specified**

Tabular Note dialog box enhancements

What is it?

You can now create and edit tabular notes from a block style dialog box.

In previous versions of NX, you could not edit the tabular note's default configuration until you placed it on a drawing sheet. Now as you can place the tabular note, you can use the options in the **Tabular Note** dialog box to define the note's:

- Alignment and orientation.
- Leader type and origin.
- Column and row configuration.
- Annotation style.

Preferences for tabular notes

You can set preferences for the tabular notes using the new options in the **Annotation Preferences** and **Annotation Style** dialog boxes.

You can:

- Specify how the table behaves when it exceeds its maximum size.

- Set the distance between adjacent table sections of the tabular note.
- Append a continuation note to the table.
- Set or edit the following table preferences:
    - Lock table formats to prevent end users from modifying the table format properties.
    - Lock data in the tabular note so it cannot be manually changed.
    - Display locked deleted rows in various ways.
    - Automatically update the part or object attributes, part numbers, and expression values inside the tabular note.

Where do I find it?

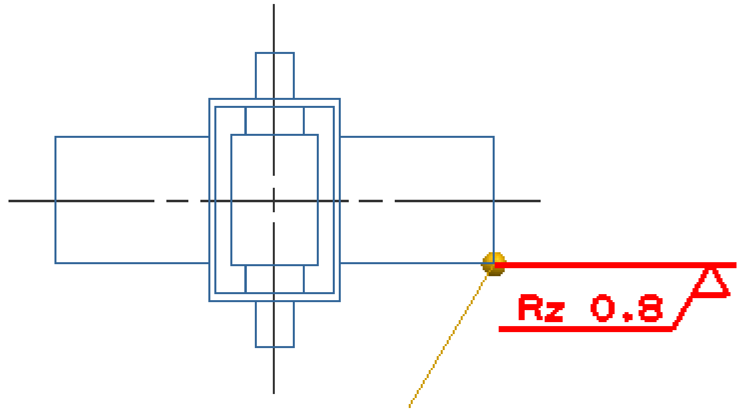
Application	Drafting
Toolbar	**Table**→**Table Drop-down** list→**Tabular Note** 
Menu	**Insert**→**Table**→**Tabular Note**
Customer defaults	**Drafting**→**Annotation**, **Tabular Note** tab or **Sections** tab.

General annotation enhancements

The following enhancements are provided to improve and optimize how you create and use annotations.

- When you create or edit a note, the **Inherit** option on the **Note** dialog box lets you inherit the text, style preference, and text alignment settings of an existing note on the current drawing sheet.
- When you position a surface finish symbol created with a leader type set to **Flag**, you can select either side of the leader line to place the symbol. The symbol will automatically change its orientation when you place it on the opposite side of the leader line.

  Note If you want any associated surface finish symbol text to also be reoriented, you must select **Invert Text** in the **Settings** group of the **Surface Finish** dialog box.



- A new **Position at Snap Point** option lets you position all annotation and symbols using the Snap Point functionality on the Selection bar.
- A new customer default, **Show All Extension Handles**, displays all centerline handles when you create or edit centerlines, regardless of the setting of the **Set Extension Individually** option.
- You can now reposition annotation that is retained due to lost associativity.
- You can add part attributes, object attributes, and non-sketch related expressions to master custom symbols.
- You can use Snap Point functionality to select points on the geometry of a custom symbol without smashing it first.

Where do I find it?

Inherit option for notes

Application	Drafting
Toolbar	**Annotation→Note**
Menu	**Insert→Annotation→Note**
Location in dialog box	**Inherit** group→**Select Note**

Surface Finish Symbol positioning

Application	Drafting
Prerequisite	The symbol must be placed with **Leader Type** set to **Flag**
Toolbar	**Annotation→Surface Finish Symbol**
Menu	**Insert→Annotation→Surface Finish Symbol**

Surface Finish Symbol positioning

Application	PMI
Prerequisite	The symbol must be placed with **Leader Type** set to **Flag**
Toolbar	**PMI→Surface Finish Symbol**
Menu	**PMI→Surface Finish Symbol**

Position at Snap Point option

Application	Drafting and PMI
Toolbar	Any command that contains the **Origin** group in the dialog box
Menu	Any command that contains the **Origin** group in the dialog box
Location in dialog box	**Origin** group® under **Alignment**® **Position at Snap Point** check box

Show All Extension Handles customer default

Application	Drafting and PMI
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Drafting**® **Annotation**® **Symbols** tab® **Show All Extension Handles** check box

Suppress stacking and alignment for new dimensions

What is it?

New **Stacking** and **Alignment** options on the **Dimensions** dialog bar let you disable automatic stacking and alignment when you add dimensions on a drawing sheet.

You can also press the ALT key to temporarily disable automatic stacking and alignment when you place dimensions.

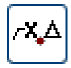
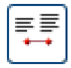
Note The **Stacking** option is available for all Drafting dimensions except ordinate, baseline, and chain dimensions. The **Alignment** option is available for all Drafting dimensions except ordinate dimensions. These options are not available while editing Drafting dimensions.

These options are not available for PMI dimensions.

Why should I use it?

Use these options to position dimensions in areas of the drawing sheet that contain a large number of annotations or other dimensions.

Where do I find it?

Application	Drafting
Toolbar	**Dimensions** dialog bar→**Stacking**  **Dimensions** dialog bar→**Alignment** 
Shortcut Menu	Right-click when you place a dimension →**Stack Annotation** Right-click when you place a dimension →**Align Horizontal or Vertical**

Crosshatch and Area Fill enhancements

The following enhancements will help you create and update complex crosshatch and area fill objects.

- Improved update behavior for crosshatch and area fill boundaries associated to section edges, extracted edges, and silhouette curves lets you more successfully update these types of annotations.

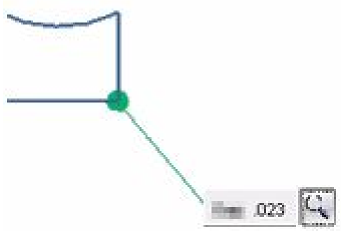
- **Find Apparent Intersections** is a new customer default you can use to create boundaries using apparent intersections; apparent intersections are intersection curves that appear to exist based on the view orientation, but do not physically exist in the geometry.

- You can use the new **Boundary Curve Tolerance** option to set how closely NX approximates the crosshatch boundary along irregular curves, such as splines and conics.

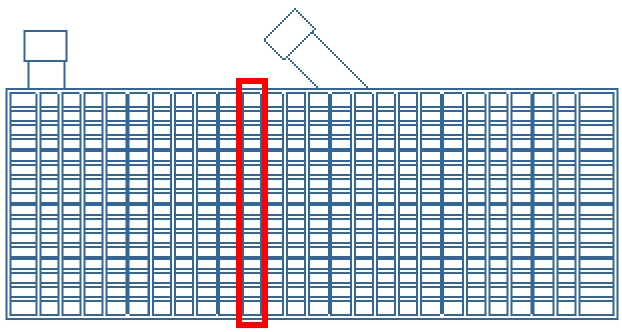
- You can use the new **Distance Tolerance** option to control the allowable gap between the boundary curves used for closed boundary detection. In Boundary Curve selection this tolerance is also used for curve rule selection.

  Note This option is available when you create or edit new boundaries only. It cannot be applied to existing boundaries.

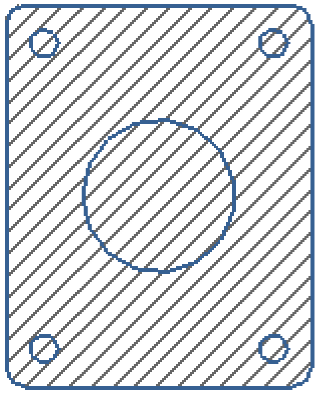
- The following options are available when the **Selection Mode** is set to **Point in Region**.
    - o **Identify Gaps** lets you detect gaps in the boundary curves, which can lead to bleeding of a crosshatch or area fill. Use the **Smaller Than** option to control the size of acceptable gaps.



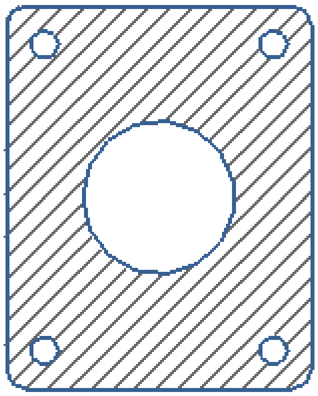
    - o The **Region to Search** options can be used to create a boundary box around the set of curves you want to process for closed boundary detection. These options are useful if you want to reduce the closed boundary detection time for drawings with a large number of curves or edges.



- **Ignore Inner Boundaries** option that lets you ignore islands and holes when you create crosshatch or area fill symbols.



Ignore Inner Boundaries selected



Ignore Inner Boundaries not selected

- You can now preview a crosshatch or area fill symbol before creating it.
- You can now see visual representations of crosshatch pattern in the dialog box for each crosshatch and area fill pattern type.
- The display of crosshatch and area fill symbols created on arrangements, explosions, deformed parts, and on geometry with assembly operations that reposition components, such as Move Component, is improved.

Where do I find it?

Distance Tolerance option

Application	Drafting
Toolbar	**Annotation® Crosshatch** **Annotation® Area Fill**
Menu	**Insert® Annotation® Crosshatch** **Insert® Annotation® Area Fill**
Location in dialog box	**Boundary** group® **Tolerance® Distance Tolerance**

Boundary Curve Tolerance option

Application	Drafting
Toolbar	**Annotation® Crosshatch** **Annotation® Area Fill**
Menu	**Insert® Annotation® Crosshatch** **Insert® Annotation® Area Fill**
Location in dialog box	**Settings** group® **Boundary Curve Tolerance** box

Crosshatch and Area Fill Boundary Curve Tolerance annotation preference

Application	Drafting
Toolbar	**Annotation® Annotation Preferences**
Menu	**Preferences® Annotation**
Location in dialog box	**Fill/Hatch** tab

Identify Gaps and **Smaller Than** options

Application	Drafting
Toolbar	**Annotation® Crosshatch** **Annotation® Area Fill**
Menu	**Insert® Annotation® Crosshatch** **Insert® Annotation® Area Fill**
Location in dialog box	**Boundary** group® **Gaps**

Region to Search options

Application	Drafting
Toolbar	**Annotation® Crosshatch** **Annotation® Area Fill**
Menu	**Insert® Annotation® Crosshatch** **Insert® Annotation® Area Fill**
Location in dialog box	**Boundary** group® **Region to Search**

Ignore Inner Boundaries option

Application	Drafting
Toolbar	**Annotation® Crosshatch** **Annotation® Area Fill**
Menu	**Insert® Annotation® Crosshatch** **Insert® Annotation® Area Fill**
Location in dialog box	**Boundary** group® **Ignore Inner Boundaries**

PMI

Model view UI improvements

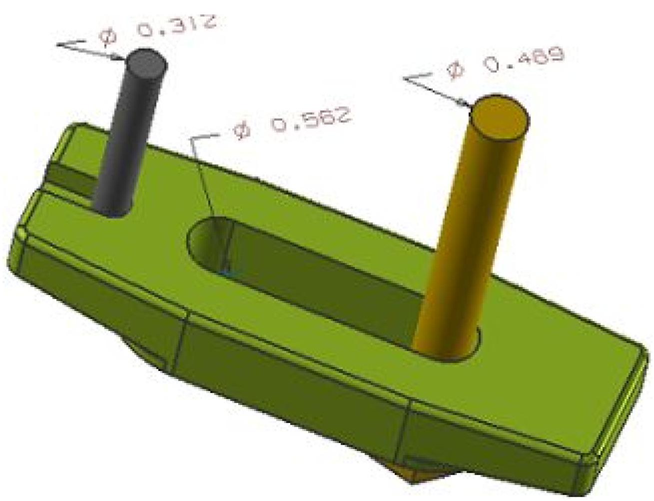
Naming convention for model view names

User defined views can now be saved in lower case letters. View names in legacy parts will still have capital letters until the name is changed by the user.

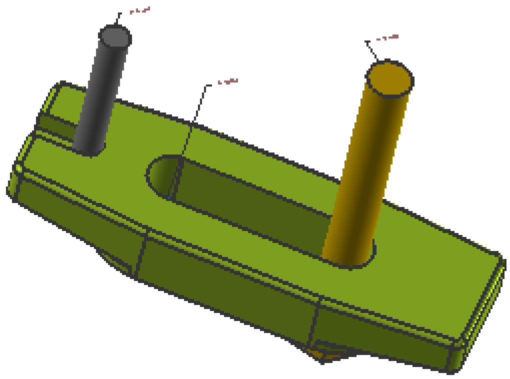
Resize PMI

Previously the size of the PMI was fixed and was controlled by the attributes for preference and style. As you zoomed the display in or out, the PMI got larger or smaller; this could cause the PMI to be too small or too large to be read on the screen.

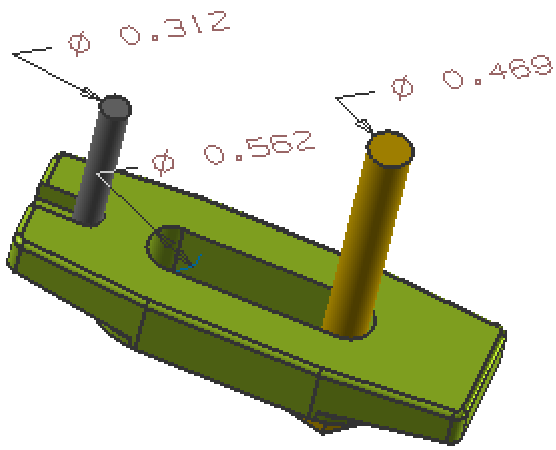
The **Resize PMI**  command enables you to resize the PMI display to make it more readable relative to the current viewing perspective.



View display before zoom



View display after zoom, PMI objects difficult to read



View display after selecting Resize PMI

You can use the resize options on the **Customer Defaults** and **PMI Preferences** dialog boxes to control the size of PMI display. These options are part preferences and are saved with the part.

Resize on Create	Enables the automatic resize of PMI at the time that it is created. This is in contrast to creating the PMI, then selecting **Resize** to adjust the size.
Resize on View Save	Determines whether or not PMI will be resized when you save a view.
Style Settings Relative to Saved View Scale	Resizes PMI based on its display attributes relative to the Saved View Scale. This option preserves the different sizes of the PMI.
Style Settings Relative to View Zoom Factor	Resizes PMI based on its display attributes relative to the **View Zoom Factor**. This option preserves the different sizes of the PMI.
Independent of Style Settings	Resizes all PMI the same size, regardless of the display attributes for each PMI.

Where do I find it?

Resize PMI

Application	PMI
Toolbar	**PMI→Resize PMI**
Menu	**PMI→Resize**
Resource bar	In the **Part Navigator**, right-click the model work view → **Resize PMI In View**

PMI Preferences

Menu	**Preferences→PMI→PMI→ Display** tab → **Resize Options**

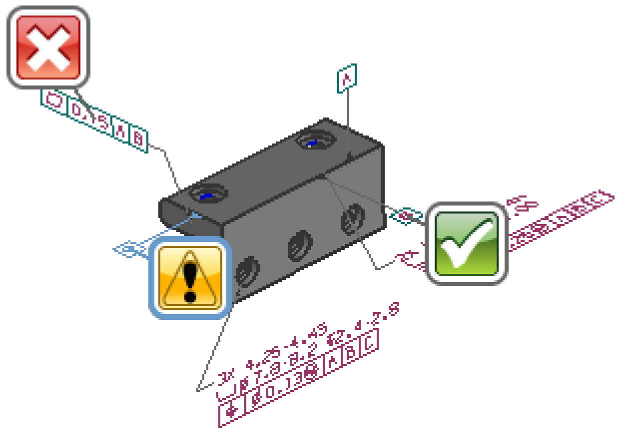
Customer Defaults

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**PMI →PMI General →Display** tab**→Resize Options**

Checking PMI GD&T Validity

You can use the two Check-Mate tests, **Check PMI GD&T Semantic Validity** and **Check PMI GD&T Syntactic Validity**, to verify that the PMI GD&T on the part is compliant with the GD&T standards for the part.

The results from the check of PMI GD&T symbols are listed in the **HD3D Tools** navigator in the **Results** group; associated visual tags also appear in the graphics window.



For more detailed information about each result, you can right-click the object in the **Results** list and select **Show Info View**.

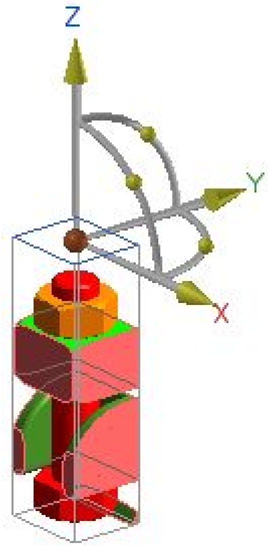
Where do I find it?

Application	PMI
Toolbar	**Check-Mate→Set Up Tests**
Menu	**Analysis→Check-Mate→Set Up Tests**
Location in dialog box	**Tests** tab→**PMI→Check PMI GD&T Semantic Validity** or **Check PMI GD&T Syntactic Validity**
Resource bar	**HD3D Tools** tab →**Check-Mate→Settings** group→**Set Up Tests**

Box type PMI Lightweight Section View

PMI lightweight section views can be defined using a box type of clipping boundary.

Only the material from selected components or bodies within the box clipping boundary will remain displayed. Components or bodies that have not been selected are unaffected. This allows for large complicated models to be simplified in order to make views of them more manageable.



Before lightweight section view **Box type section view**

Existing PMI lightweight section views will be converted to a one plane type of PMI lightweight section view.

How do I use it?

To create a Box PMI lightweight section view, select the desired type on the PMI lightweight section view dialog box and use the controls on screen and on the dialog to define the placement of the clipping planes.

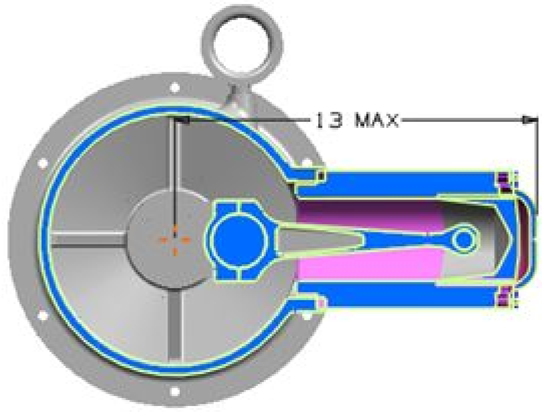
Where do I find it?

Application	PMI
Toolbar	**PMI→Lightweight Section View**
Menu	**PMI→Section→Lightweight Section View**
Location in dialog box	**Lightweight Section View** dialog box ® **Type** list ® **Box**

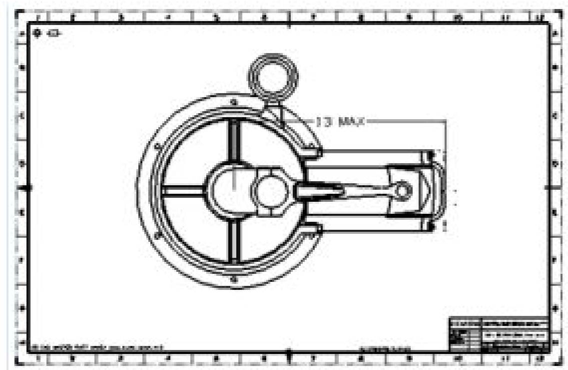
Inheriting PMI lightweight section views on drawings

You can now place a PMI lightweight section view on a drawing as a base view.

The inherited PMI lightweight section view is an associative copy of the 3D PMI Model View, and any changes to the section view need to be made in the 3D PMI Model View.



A one plane type lightweight section view



The same view inherited on a drawing

New customer default options for inherited lightweight section views

Crosshatch Inherited PMI Lightweight Section Views — Lets you determine whether to add crosshatching to the cut face portions of a PMI lightweight section view that has been inherited to a drawing.

Compound Box Type Lightweight Section View — Lets you determine whether multiple inherited box type PMI lightweight section views can be combined into a drafting view.

Where do I find it?

Application	Drafting
Toolbar	**Drawing→Base View** 
Menu	**Insert→View→Base**
Location in dialog box	**Model View** group® **Model View to Use**

Customer defaults

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Drafting→View→General** tab →**Crosshatch Inherited PMI Lightweight Section Views** option **Drafting→General→View** tab →**Compound Box Type Lightweight Section View** option

Crosshatch Inherited PMI Lightweight Section Views

Application	Drafting
Menu	**Preferences→View**
Location in dialog box	**Inherit PMI** tab →**Crosshatch Inherited PMI Lightweight Section Views**

PMI Search enhancements

You can now use the **PMI Search**  command to find datum feature symbols and datum targets on a model that meet the criteria you specify. You can examine PMI symbols that match the criteria one by one, or you can save the results to a search model view, which you can review at a later time. Search model views are saved with the part.

The following datum target types are supported:

- Point
- Line
- Rectangular
- Circular
- Annular
- Cylindrical
- Arbitrary

Where do I find it?

Application	PMI
Toolbar	**PMI→PMI Search** 
Menu	**Information→PMI→Search**

WAVE PMI Linker

Use the **WAVE PMI Linker** to make an associative copy of a PMI object from one part to another so that other operations, such as Manufacturing, can reference the PMI. When the contents of the source PMI are changed, the PMI in the target part will also be updated. You can select the target part and the source part from which the PMI will be copied.

Geometry to WAVE Link

You can select either Body or Topology to WAVE link.

- **Body** – NX will prefer to WAVE link the entire solid body that contains the edge/face to which the PMI is associated.
- **Topology** – NX will prefer to WAVE link the edge/face as a separate WAVE link feature.

By creating a WAVE link for the PMI object, you can track the relationships of the object between parts using the Assembly WAVE Interpart Link Browser or the Relations Browser.

Note If you only want to see component PMI in a parent assembly, you can use the PMI Assembly Filter instead of making an associative copy of the PMI.

Where do I find it?

Application	PMI
Menu	**Insert→Associative Copy→WAVE PMI Linker**

Support for Standard Fonts

Standard font types available in the FreeType Font library are supported in the PMI and Drafting environments.

Note Although the FreeType Font library supports non scalable and other font types, only scalable TrueType, OpenType, and PostScript fonts are supported.

When setting your character font, it is important to remember the following points:

- Existing NX font types are still available and are located in a directory set by the **UGII_CHARACTER_FONT_DIR** environment variable.
- Standard Fonts files are found in your normal font directories, usually in the *C:\Windows\Fonts* on Windows or in a configuration file on Unix.
- You can use custom fonts by placing the font files in a directory referenced by the **UGII_STANDARD_FONT_DIR** environment variable.

- NX will search for the character font in the following order:
  1. Standard font location (that is, *C:/Window/Fonts*).
  2. The directory specified by **UGII_STANDARD_FONT_DIR**.
  3. The directory specified by **UGII_CHARACTER_FONT_DIR**.
- A default font is used when a font cannot be found. The default font can be changed, but is initially set to Arial Unicode MS. If Arial Unicode MS is not found, the default font is set to Tahoma. If Tahoma is not found, then the default font is set to Arial.

Why should I use it?

Use Standard Fonts to replace, enhance, or supplement your current set of drafting and PMI character fonts.

Where do I find it?

All fonts found in standard directories and in the custom directory are automatically available in customer defaults, preference dialog boxes, style dialog boxes, and any other dialog box that lets you specify a character font.

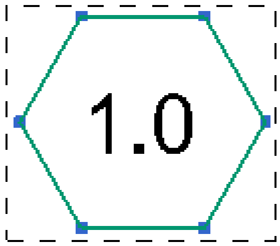
Associative custom symbols

When you create custom symbols from a master custom symbol definition, you can make the custom symbols associative to the master custom symbol. If the master custom symbol changes, the associative copies of the symbol also change.

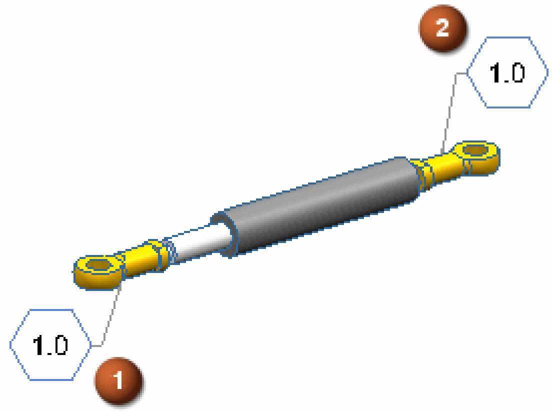
Note Associativity is not supported for nested custom symbols or PMI custom symbols inherited onto a drawing.

When you create associative custom symbols, it is important to note the following.

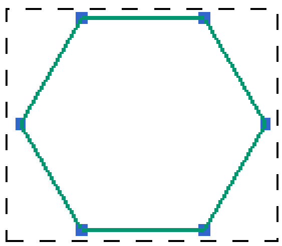
- Associativity to the master symbol is lost if the master symbol is deleted, the folder containing the master symbol is deleted, or the symbol instance is smashed into its constituent parts. You cannot add master symbol associativity to existing symbol instances created in previous NX releases.
- When you create an instance of a master custom symbol, the **Lock Update** option is available. When set, the symbol instance is locked, and changes to the master symbol definition do not affect the instance.



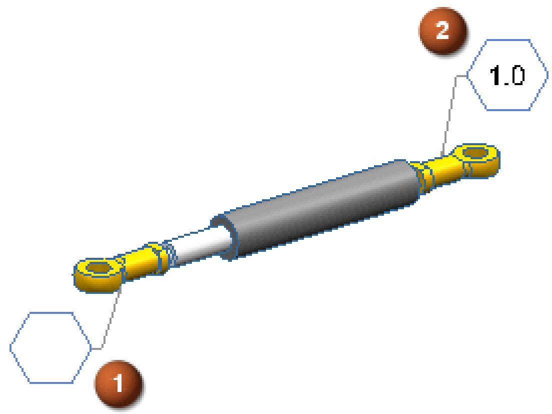
Original master symbol



Symbol instances from original master symbol



Changed master symbol



Symbol instances from changed master symbol

1 – Unlocked symbol 2 – Locked symbol

- You can manually update symbol instances that are unlocked.

  See **Out-of-Date** Folder for additional information.

- You can also change the locked status of existing custom symbol instances.

- The **Reassociate** command lets you reassociate a symbol instance to its master symbol definition. You can only use the **Reassociate** command on symbol instances created in NX 8 and above.

Create Associative Symbols customer default and the Lock Update option

A new customer default, **Create Associative Symbols**, determines whether or not to create an associative custom symbol instance. It also determines the visibility and availability of the **Lock Update** option in the **Custom**

Symbol dialog box. When **Create Associative Symbols** is selected, the **Lock Update** option is visible and available, and you can create both associative and non-associative symbol instances. When it is not selected, the **Lock Update** option is not visible and symbol instances are always created as non-associative copies of the master custom symbol.

Why should I use it?

Create associative custom symbol instances when you want your symbol instances to always reflect the master definition of the original custom symbol.

Where do I find it?

Lock Update option

Application	Drafting and PMI
Toolbar	In the Drafting application: **Symbol**® **Symbol Drop-down** list® **Custom Symbol**  In the PMI application: **PMI**® **PMI Symbols Drop-down**® **Custom Symbol** 
Menu	In the Drafting application: **Insert**® **Symbol**® **Custom** In the PMI application: **PMI**® **Symbol**® **Custom**
Location in dialog box	**Settings** group® **Lock Update**

Create Associative Symbols customer default

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Drafting**® **Custom Symbols**® **All** tab® **Create Associative Symbols**

Reassociate command

Application	Drafting and PMI
Graphics Window	Right-click custom symbol instance® **Reassociate**

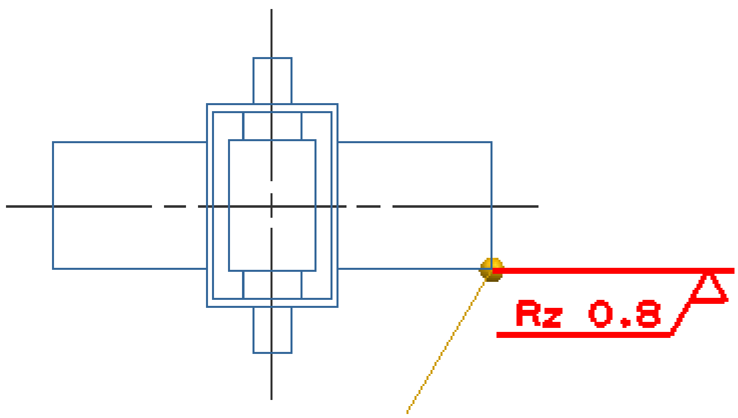
General annotation enhancements

The following enhancements are provided to improve and optimize how you create and use annotations.

- When you create or edit a note, the **Inherit** option on the **Note** dialog box lets you inherit the text, style preference, and text alignment settings of an existing note on the current drawing sheet.

- When you position a surface finish symbol created with a leader type set to **Flag**, you can select either side of the leader line to place the symbol. The symbol will automatically change its orientation when you place it on the opposite side of the leader line.

  Note If you want any associated surface finish symbol text to also be reoriented, you must select **Invert Text** in the **Settings** group of the **Surface Finish** dialog box.



- A new **Position at Snap Point** option lets you position all annotation and symbols using the Snap Point functionality on the Selection bar.

- A new customer default, **Show All Extension Handles**, displays all centerline handles when you create or edit centerlines, regardless of the setting of the **Set Extension Individually** option.

- You can now reposition annotation that is retained due to lost associativity.

- You can add part attributes, object attributes, and non-sketch related expressions to master custom symbols.

- You can use Snap Point functionality to select points on the geometry of a custom symbol without smashing it first.

Where do I find it?

Inherit option for notes

Application	Drafting
Toolbar	**Annotation→Note** 
Menu	**Insert→Annotation→Note**
Location in dialog box	**Inherit** group→**Select Note** 

Surface Finish Symbol positioning

Application	Drafting
Prerequisite	The symbol must be placed with **Leader Type** set to **Flag**
Toolbar	**Annotation→Surface Finish Symbol** 
Menu	**Insert→Annotation→Surface Finish Symbol**

Surface Finish Symbol positioning

Application	PMI
Prerequisite	The symbol must be placed with **Leader Type** set to **Flag**
Toolbar	**PMI→Surface Finish Symbol** 
Menu	**PMI→Surface Finish Symbol**

Position at Snap Point option

Application	Drafting and PMI
Toolbar	Any command that contains the **Origin** group in the dialog box
Menu	Any command that contains the **Origin** group in the dialog box
Location in dialog box	**Origin** group® under **Alignment**® **Position at Snap Point** check box

Show All Extension Handles customer default

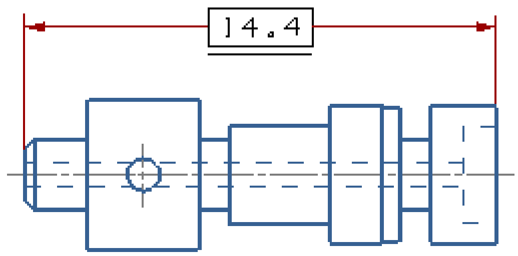
Application	Drafting and PMI
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Drafting**® **Annotation**® **Symbols** tab® **Show All Extension Handles** check box

ASME Y14.5-2009 Drafting Standard enhancements

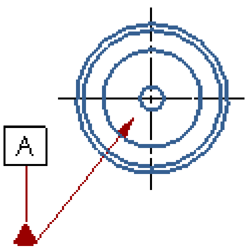
What is it?

New options are available to create ASME Y14.5-2009 compliant annotations. Enhancements to the Drafting and PMI annotation functionality let you do the following:

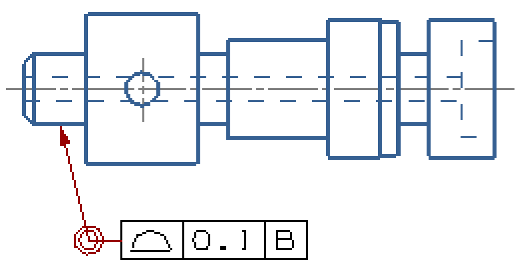
- Create basic dimension types that are not to scale.



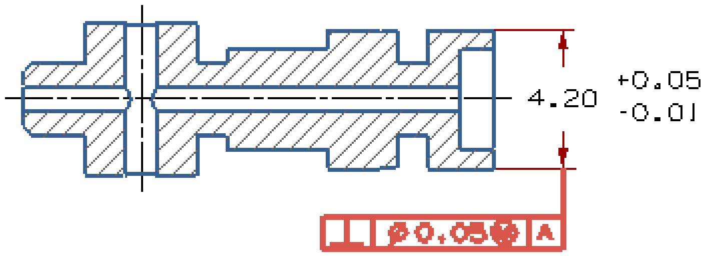
- Create a **Datum on Dot Terminated** leader with an arrowhead for Feature Control Frames (FCS), Datum Feature Symbols (DFS), and notes.



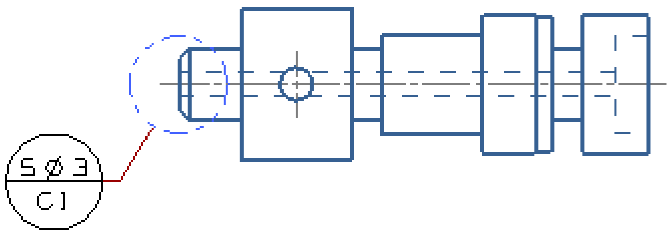
- Create a leader with an all over symbol for a feature control frame.



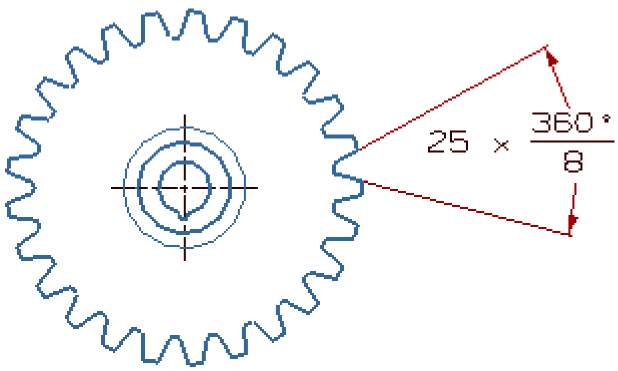
- Correctly attach annotations, such as a feature control frame, to a dimension using a **Flag** leader.



- Create a datum target symbol with a spherical zone shape.



- Specify an angular dimension as a fractional division of an arc.



Where do I find it?

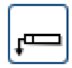
Basic, Not to Scale dimension type

Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation® Annotation Preferences**
Menu	In the Drafting and PMI applications: **Preferences® Annotation**
Location in dialog box	**Dimensions** tab® Dimension Display Type list® **Basic, Not to Scale**

Datum on Dot Terminated leader with an arrowhead

Application	Drafting
Toolbar	**Annotation→Datum Feature Symbol** **Annotation→Feature Control Frame** **Annotation→Note**
Menu	**Insert® Annotation® Datum Feature Symbol** **Insert® Annotation® Feature Control Frame** **Insert® Annotation® Note**
Location in dialog box	**Leader** group® **Type® Dot Terminated** **Style** subgroup® **Arrowhead**

Leader with an **All Over** symbol for a feature control frame

Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation® Feature Control Frame**  In the PMI application: **PMI® PMI Annotation Drop-down® Feature Control Frame** 
Menu	In the Drafting application: **Insert® Annotation® Feature Control Frame** In the PMI application: **PMI® Feature Control Frame**
Location in dialog box	**Leader** group® **Type® All Over**

Datum target symbol with a spherical zone shape

Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation® Datum Target**  In the PMI application: **PMI® Datum Target** 
Menu	In the Drafting application: **Insert® Annotation® Datum Target** In the PMI application: **PMI® Datum Target**
Location in dialog box	**Type** group® **Spherical**

Angular dimension as a fractional division of an arc

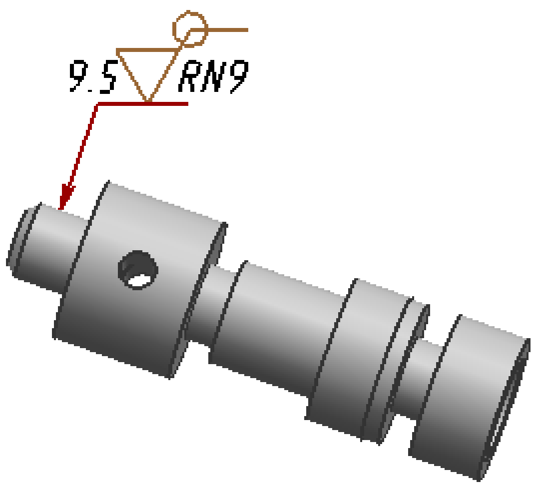
Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation®** **Annotation Preferences** 
Menu	In the Drafting and PMI applications: **Preferences®** **Annotation**
Location in dialog box	**Units** tab® **Display as a Fraction**

ESKD Drafting Standard enhancements

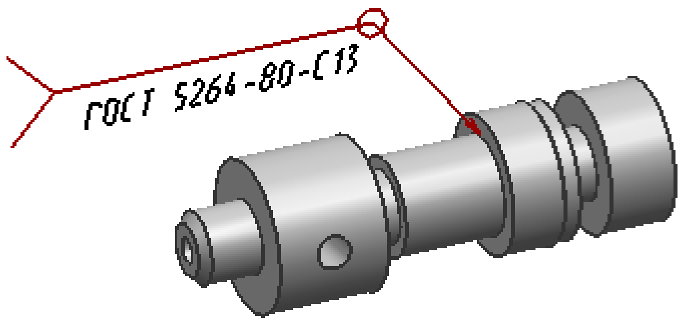
The following enhancements are provided in both Drafting and PMI to better support the ESKD drafting standard.

Note The ESKD standard must be set in customer defaults, or in **Surface Finish** option on the **Symbols** tab of the **Annotation Preferences** dialog box.

- You can specify surface finish symbols with roughness values in which the lay symbol and roughness are in a single line.



- The **Not Specified** weld type lets you create a symbol with just weld properties and no specific weld type.



Where do I find it?

Surface finish symbol with a single line roughness callout

Application	Drafting and PMI
Prerequisite	Your drafting standard must be set to **ESKD** in the **Annotation Preferences** dialog box.
Toolbar	In the Drafting application: **Annotation® Surface Finish Symbol** In the PMI application: **PMI® Surface Finish**
Menu	In the Drafting application: **Insert® Annotation® Surface Finish Symbol** In the PMI application: **PMI® Surface Finish® Attributes** group® **Standard® ESKD**
Location in dialog box	**Attributes** group® **Single Roughness Value**

Not Specified weld type

Application	Drafting and PMI
Prerequisite	Your drafting standard must be set to **ESKD** in the **Annotation Preferences** dialog box.
Toolbar	In the Drafting application: **Annotation® Weld Symbol** In the PMI application: **PMI® Weld Symbol**
Menu	In the Drafting application: **Insert® Annotation® Weld Symbol** In the PMI application: **PMI® Weld Symbol**
Location in dialog box	**Other Side** group® Weld Symbol list® **Not Specified** **Arrow Side** group® Weld Symbol list® **Not Specified**

PMI support in STEP translator

What is it?

You can now translate PMI data from NX to STEP as polyline (presentation) data. You can also now import PMI (polyline form) from STEP to NX.

Why should I use it?

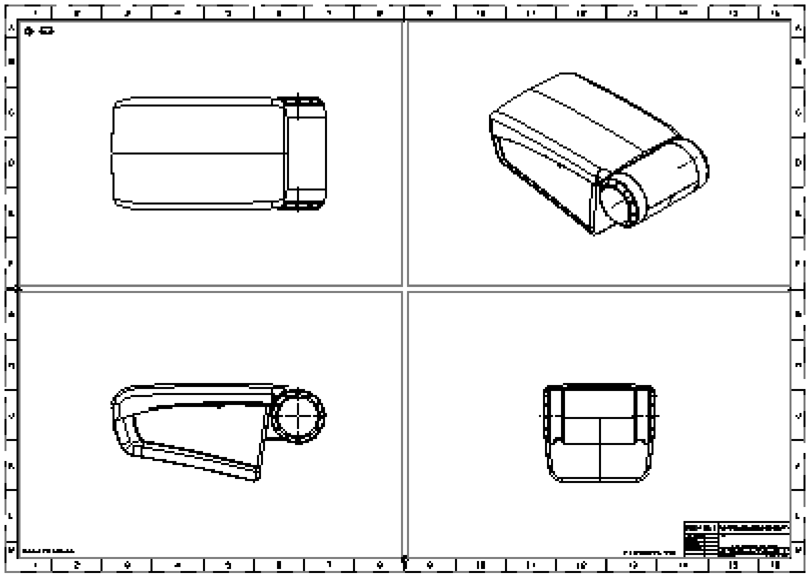
Use this if you want to translate NX PMI to STEP and **Import** STEP PMI to NX.

Where do I find it?

Application	NX
Menu	**File® Import® STEP203** **File® Import® STEP214** **File® Export® STEP203** **File® Export® STEP214**

View Creation Wizard

The **View Creation Wizard** simplifies the process of adding one or more drafting views to a drawing sheet.



The wizard steps you through the following process of:

- Selecting a currently loaded, recently loaded, or unloaded part or assembly.
- Specifying the preview style and view display parameters.
- Optionally inheriting existing PMI data from the assembly model.
- Optionally selecting existing assembly arrangements.
- Specifying a view orientation for the parent view.
- Constructing a multi-view layout.

Note The **View Creation Wizard** is available for master model drawings only. It is not available for standalone drawings.

By default, the **View Creation Wizard** is launched immediately after creating a new drawing or a new drawing sheet. You can change this default behavior by adjusting the options on the **General** tab of the **Drafting Preferences** dialog box, or by adjusting the options on the **Workflow** tab under **Drafting→Drawing** on the **Customer Defaults** dialog box.

The View Creation Wizard window

The options in the **View Creation Wizard** window step you through the process of defining and placing a group of views on the current drawing sheet.

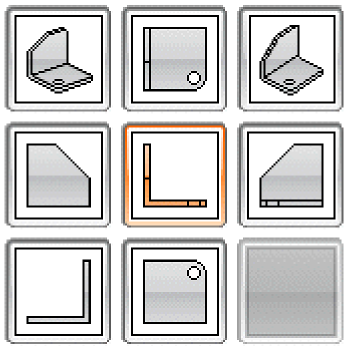
You can:

- Select the part or assembly to be depicted on the drawing.

- Set your preferences for the display. You can click **View Style** to set view style preferences not directly available from the from the **View Creation Wizard** window.

- Specify the parent view orientation.

- Configure a view layout. You can include up to six standard orthographic views in the layout, along with an isometric or trimetric view. The projection angle of the views is derived from the projection angle of the drawing.



- Place the view layout on the drawing sheet using **Margins** to define the minimum distance between adjacent view borders in the layout, and between view borders and the edge of the drawing sheet.

If the **View Placement** option is set to **Automatic**, you can complete the placement of the views in the first step. Use the **Manual** option if you want to place the center of the view layout anywhere on the drawing sheet.

Where do I find it?

Application	Drafting
Toolbar	**Drawing→Add View Drop-down** list® **View Creation Wizard** **Drawing→View Creation Wizard**
Menu	**Insert→View→View Creation Wizard**

Sheet dialog box:

Location in dialog box	**Settings** group® **Automatically Start View Creation**® **View Creation Wizard**

View Creation Wizard preferences:

Menu	**Preferences→Drafting**
Location in dialog box	**General** tab® **Model–based Drawing Workflow**® **View Creation Wizard**

View Creation Wizard customer defaults:

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Drafting**® **General**® **Drawing** tab® **Model–based Drawing Workflow**® **View Creation Wizard**

Shape Studio

Highlight Lines enhancements

What is it?

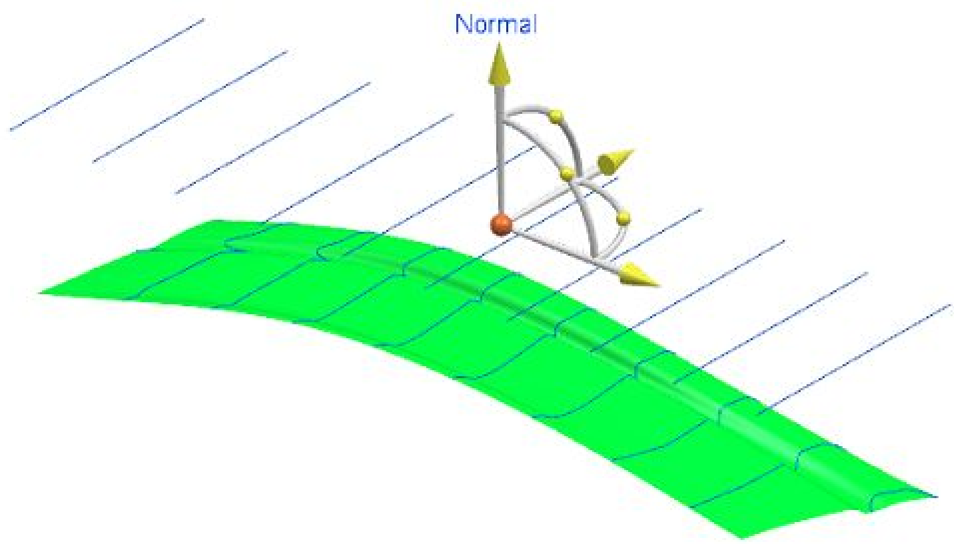
The enhancements to the **Highlight Lines** dialog box makes it easier to do the following:

- You can use the **Light Placement** option list to specify how the lights are configured.

- You can use the **Point Dialog** button to specify locations when using the **Through Points** or **Between Points** placement method.

- You can use the **Number of Lights** and **Light Spacing** options or the input fields.

In addition, new **Light Plane** options that work in conjunction with the **Manipulator** enhance your ability to designate and manipulate the light plane.



- You can choose from principal **XC**, **YC**, **ZC** planes. An **Offset** option is available when using these options.
- You can choose to designate an **Arbitrary Plane** using standard **Specify Plane** options.
- You can specify the display resolution of the analysis.

Where do I find it?

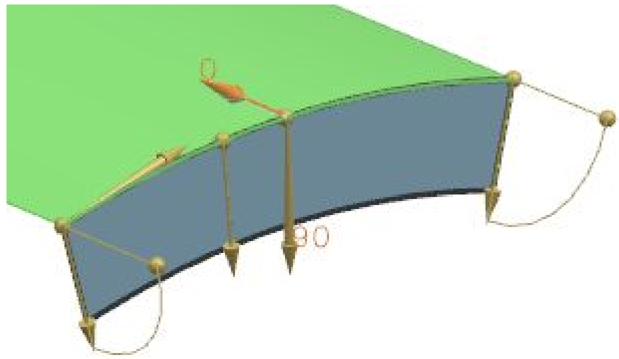
Application	Modeling and Shape Studio
Toolbar (Shape Studio only)	**Analyze Shape® Highlight Lines** 
Menu	**Analysis® Shape® Highlight Lines**

Law options enhancements

What is it?

Commands that have Law options have been enhanced in the following ways.

- You can now use dynamic manipulation handles for law types **Constant**, **Linear** and **Cubic**.



- You can use shortcut menus for parameter editing for these law types:
  - **Constant**
  - **Linear**
  - **Cubic**
  - **Multi-transitional**
  - **Non-inflecting**
  - **S-shaped**

- The dialog box options for **S-shaped** and **Non-inflecting** law types have been consolidated and more descriptive labels for these options have been added.

- Handles applied between start and end nodes have an initial **Transition** value of **Free**.

Commands that use the **Multi Transition** law have been updated to eliminate possible overlapping Law definitions.

The **Linear Along Spine** and **Cubic Along Spine** laws are removed because the **Multi Transition** laws **Linear** and **Blend** provide similar functionality.

The following commands are affected:

- **Aesthetic Face Blend**
- **Law Extension**
- **Silhouette Flange**

Where do I find it?

Application	Modeling and Shape Studio
Commands	**Offset Curve** **Law Extension** **Section Surface** **Swept** **Aesthetic Face Blend** **Silhouette Flange** **Global Shaping**
Toolbars	**Curve** **Feature** **Surface**
Menu	**Insert**® **Curve from Curves**® **Offset Curve** **Insert**® **Detail Feature**® <various commands> **Insert**® **Mesh Surface**® **Section** **Insert**® **Sweep**® **Swept** **Insert**® **Flange Surface**® <various commands>

Draft analysis objects

What is it?

The **Draft Analysis** command now creates a draft analysis object (DAO) when you click **OK** or **Apply** in the dialog box.

- Draft analysis now supports facet bodies.
- Draft analysis objects appear as objects in the **Part Navigator**.
- You can select multiple faces and facet bodies for the analysis, but only one analysis object is created.
- You can have multiple analysis objects on a single face or facet body.

The **Draft Analysis** command also has these enhancements:

- A color legend appears for each displayed analysis object, indicating areas that are:
  - o Above draft
  - o Above draft limit
  - o Below draft limit
  - o Below draft
- The display mode switches automatically to **Face Analysis** when you click the command.
- You can optionally create non-associative isocline curves from the analysis.
- You can reverse the normals of all input faces and facet bodies in a single operation or individually one by one.
- You can create labels to visualize local draft angle values and drag them across the selected faces.
- You can specify the display resolution of the analysis.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Analyze Shape® Draft Analysis** 
Menu	**Analysis® Shape® Draft**

Snip Surface enhancements

What is it?

The functionality of the **Isoparametric Trim/Divide** command is consolidated with the **Snip Surface** command.

You can use the **Snip Surface** command to do the following:

- Snip along surface isoparameters.
- Snip with a surface.
- Edit a copy of the original surface.
- View a deviation display of the snipped surface from the original.

The **Isoparametric Trim/Divide** command is no longer available.

Note You can trim using UV parameters with the **Enlarge** command. This functionality is not duplicated in the **Snip Surface** command.

Why should I use it?

You can now perform divide operations with the **Snip Surface** command to produce single unified Class-A surfaces with a reduced number of workflow steps.

Where do I find it?

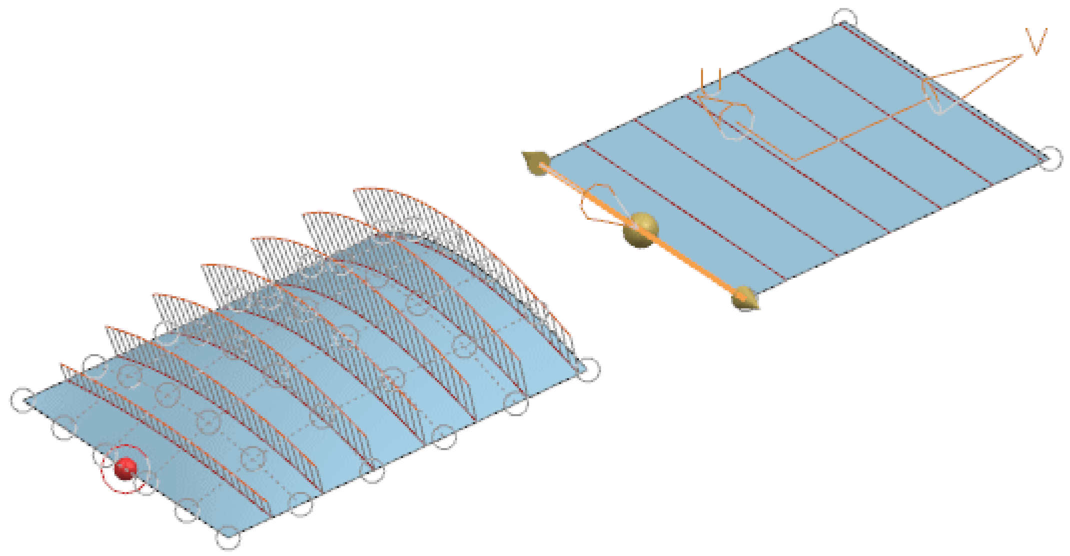
Application	Modeling, Shape Studio
Toolbar	**Edit® Snip Surface** 
Menu	**Edit® Surface® Snip Surface**

Match Edge enhancements

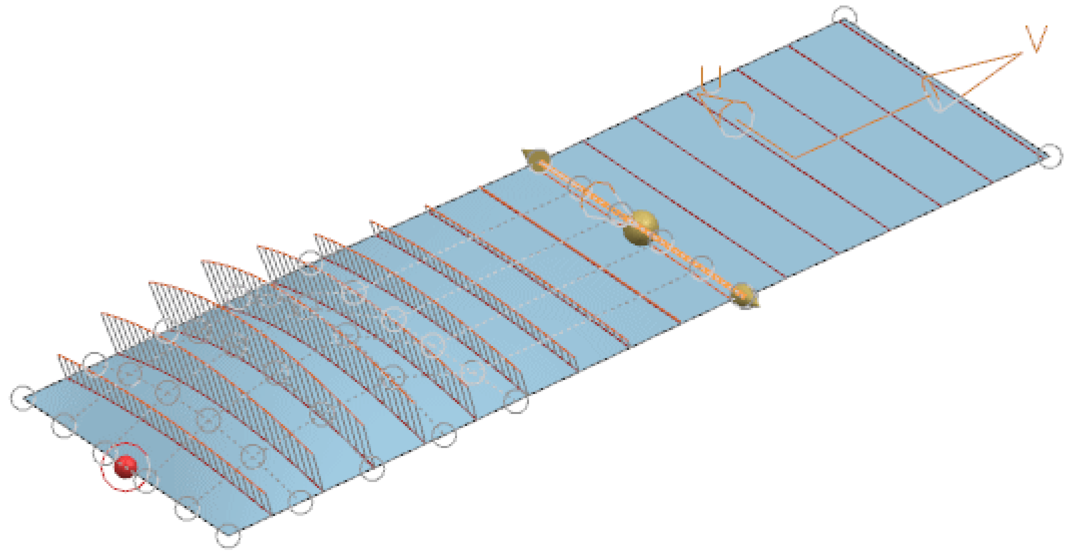
What is it?

The **Match Edge** command has been enhanced with the following functionality:

- You now can edit poles using **Shape Control** and then use the **Movement** options without having to click **Apply** after you edit the poles.
- In the **Method** group, the **Opposite Edge Constraint** option is now outside the **Movement** and **Shape Control** tabs so you can set this option in conjunction with the options on either tab.
- You can now use **Edit Poles** when in **Associative Edit Freeform** mode.
- When in **Associative Freeform Editing** mode, you can select the **Keep Selected** check box in the **Settings** group to keep the **Edge to Edit** and **Reference** objects selected for subsequent match edge operations.
- Pole structure can now be displayed on both the **Edge to Edit** and **Reference** faces.
- The **Reset Edit Poles** option in **Shape Control** resets any poles that were edited to their original state.
- When using **Section Analysis** with **Match Edge**, **Preview** now shows how the analysis object will be updated based on the proposed match edge surface.



Preview – off



Preview – on

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Edit Surface** ® **Match Edge** 
Menu	**Edit** ® **Surface** ® **Match Edge**

Match Edge and Edge Symmetry enhancements

What is it?

The following are available for the **Match Edge** command with the **Project** movement method:

- **Fix Poles** options **Fix Start** and **Fix End** to constrain the position of the start and end poles.

- Offset handles that behave similar to the **Normal** movement method.

For the **Edge Symmetry** command, a **WCS** movement method has been added with **XYZ** vector options to specify direction.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Edit Surface® Match Edge** **Edit Surface® Edge Symmetry**
Menu	**Edit® Surface® Match Edge** **Edit® Surface® Edge Symmetry**
Dialog box	**Match Edge® Method** group ® **Movement** tab **Edge Symmetry** ® **Method** group ® **Movement** tab

X-Form enhancements

What is it?

The **X-Form** command has been enhanced in the following ways.

- You can now edit non-B-Spline curves using **X-Form**.
  - Selected curves are converted to B-Splines for editing. In the case of curve features, curve parameters are removed before conversion.
  - You can set your preferred default command for spline editing.

This is available in two places.

- **Preferences® Modeling® Freeform**

- **File® Utilities® Customer Defaults® Modeling® Freeform Modeling**

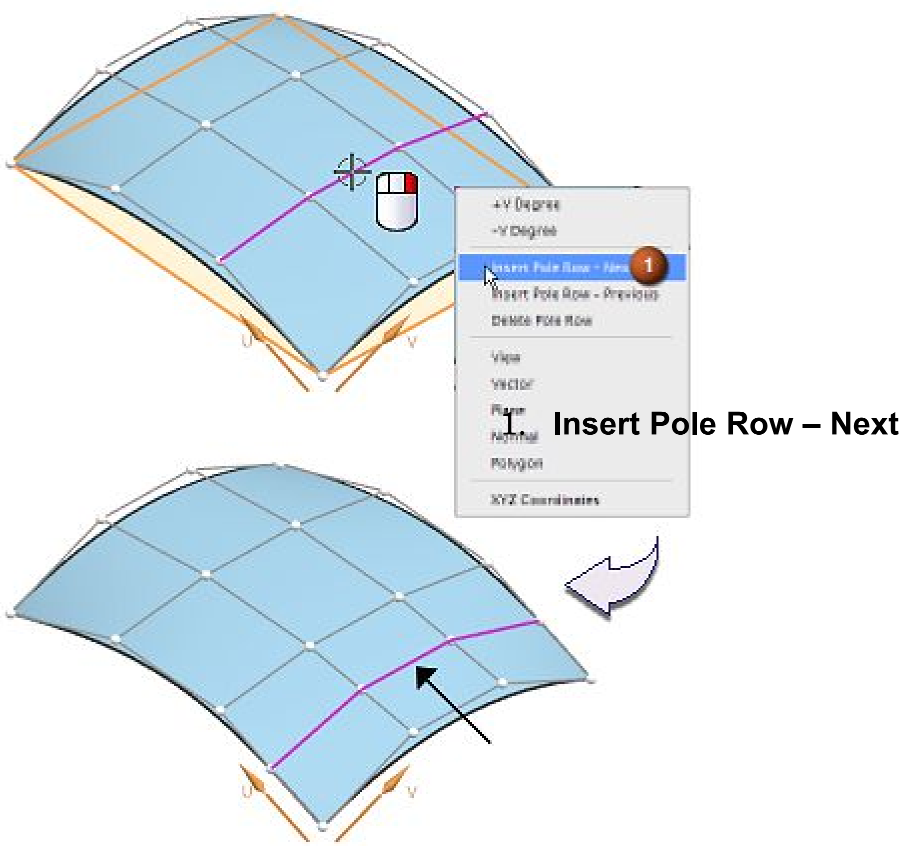
- You can now use the keyboard arrow keys, to step up and step down the input **Step Value** when moving poles in conjunction with **Microposition** options.

  o Arrow-up - key corresponds to the **+** button.

  o Arrow-down ¯ key corresponds to the **–** button.

- You can right-click a polyline to insert a row of poles before or after the selected polyline.

  When a row of poles is inserted, the original rows of poles are not displaced and maintain their original position.



- When you are using the Shape Studio application, you can double-click on unparameterized surfaces to start the **X-Form** command.

- You can now move one pole or set of poles by specifying X, Y, and Z coordinates.
  - o The **XYZ** option is now available when using the **WCS** option in the **Move** tab.
  - o **XYZ Coordinates** can also be accessed from the shortcut menu.
  - o You specify X, Y, Z coordinates using an on-screen input box.
- You can now use Shift-select to deselect an input face and select another face during **X-Form** editing.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Edit Surface® X-Form**
Menu	**Edit® Surface® X-Form**

Aesthetic Face Blend enhancements

What is it?

The **Aesthetic Face Blend** command has been enhanced in the following ways.

- You can now specify a Spine Curve when creating a Chord Length Tangent Line type blend.

  Specifying a spine curve helps orient the cross sections of the blend normal to the specified spine curve, which helps improve the output blend surface.

- In cases where the continuity of input faces is not within a specified tolerance, you can use **Blend Across Sharp Edges** in the **Settings** group to compute (or modify) the tangent hold lines of the blend.
- You can now specify a variable chord length for tangent line creation in addition to the current constant chord length tangent line control.
- In blend cases where the rolling ball radius is larger than the curvature radius of the input face chains and the blend washes out, use the **Washout Surface Creation** option to build a washout surface patch.
- You can use the **Overriding Trim Object** option to manually specify the start and end of the blend surface.
- You can now specify G3 continuity constraints on conic shaped cross sections while holding the center radius specification.

- You can now control the Tangent Magnitude on each input face chain independently.
  - o The tangent magnitude uses a constant law value on each face chain along the length of the blend.
  - o You can specify tangent magnitude using either on-screen handles or the input fields in the dialog box.
- You can now specify separate, individual continuity constraints on each face of an input face chain. For instance, the continuity of the first input chain of faces can be G1, while the second input chain of faces can have G2 continuity.
- The **Sew All Faces** option has been added to **Trim Options**.
- You can now trim a blend at a specified location, which lets you limit the blend to a selected object such as a plane, face or edge.
- You can use the **Non-associative Center line Curve Output** option specify a non-associative centerline curve of the blend be created when the blend is generated.
- Use the **Bezier Output** option in the **Settings** group to specify Bezier surface patches be generated instead of B-surfaces. The **Bezier Output** option divides blend faces at knot values.

  Note the **Trim and Attach** option is not available with Bezier output.

- You now can control the segmentation of a generated blend surface. There are four options.
  - o Use the **Both Face Chains** option to segment the blend surface at the boundaries of both the input face chains.
  - o Use the **On Face Chain 1** option to segment a blend surface at all the boundaries of first face chain.
  - o Use the **On Face Chain 2** option to segment a blend surface at all the boundaries of second face chain.
  - o Use the **On Both Face Chains and Merge Small Blend Faces** option to segment the blend surface at all the face chains then merge all the faces whose tangent curves are shorter than a specified threshold value.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Feature® Aesthetic Face Blend**
Menu	**Insert® Detail Feature® Aesthetic Face Blend**

Data Reuse

Reuse Library

Reuse Library Management

Use the **Reuse Library Management** command to configure the content of the Reuse Library. You can:

- Add user defined libraries to the Reuse Library, or remove them.
- Edit the descriptive name, path, and the icon of the libraries.
- Specify which libraries are visible in the Reuse Library.
- Load your own library configuration file.
- Specify the NX applications in which a library is available.
- Set the catalog visibility for the standard parts library.

You can also use this command to access libraries which are stored on your machine, even when you run Teamcenter Integration. Libraries on your machine can include the reusable object library, UDF library, and 2D section library.

Where do I find it?

Resource Bar	Reuse Library
Toolbar	**Reuse Library® Reuse Library Management**
Menu	**Tools® Reuse Library® Reuse Library Management**
Reuse Library Navigator	Right-click the title of the main panel ® **Reuse Library Management**

Integrating reusable objects with a spreadsheet

You can define values and options in a spreadsheet and integrate that spreadsheet with a reusable object or a knowledge enabled part. When you add a knowledge enabled part or a reusable object to the graphics window, the **Add Reusable Component** or the **Reuse Paste** dialog box appears. The dialog box displays the available parameter values that are listed in the spreadsheet, and lets you chose a value for each parameter.

In the spreadsheet, you can define values and options for the following:

- The location of the bitmap file used as the preview image for the object
- Part attributes
- Expressions

Expressions can be listed in primary, secondary, and tertiary levels in the spreadsheet. In this format, the values available at each level depend on the value selected for the preceding parameter or expression. For example, **A** is a primary expression with two value options of **10** and **20**. **B** is a secondary expression and the value of that can be **0** or **5** if **A=10** and **5** or **10** if **A=20**.

BITMAP	*./cylinder.bmp*
PARAMETERS	
A	**B**
10	0,5
20	5,10
END	

Where do I find it?

Resource Bar	Reuse Library 
Prerequisite	You must drag a reusable object or a knowledge enabled part from the Reuse Library to the graphics window.

Reuse Paste

When you add a reusable object to your work part from the Reuse Library, you can now use the **Reuse Paste** dialog box to do the following:

- Change the position or orientation of the part.
- Modify any parameters specified in the spreadsheet integrated with the reusable object.

- Assign a Boolean operation and Boolean sequence to the selected target body.
- Position a sketch-based reusable object with a single click.
- Associate a sketch-based reusable object to a datum CSYS.
- Edit a reusable object.

The **Reuse Paste** dialog box is available for the following reusable objects:

- Face, body, or general type
- 2D sections with spreadsheet data
- Parts with the part attribute *REUSE_LIBRARY_OBJECT_TYPE* set to *REUSE_IMPORT*

Where do I find it?

Resource Bar	Reuse Library 
Reuse Library Navigator	In the **Member Select** area, right-click a reusable object and choose **Insert**. Drag a reusable object from the **Member Select** area to the graphics window.

Reuse Library navigator enhancements

In the **Reuse Library** navigator, you can now:

- Right-click a library folder and choose **Open Source Folder** to view and open the directory path for a library.
- Copy or cut a reusable object in the **Member Select** area, and paste it to another folder in that area.
- Display an HTML document. This document must have the same name as the object in the **Member Select** area.
- Display a descriptive name for the part attribute in the **Member Select** area. You can specify the descriptive name using the **Customer Defaults** dialog box.

Where do I find it?

Resource Bar	Reuse Library
Reuse Library navigator	Main panel ® Right-click a library folder ® **Open Source Folder** **Member Select** panel ® Right-click an object ® **Copy** or **Cut**

Reusable object examples

NX now provides examples for the following types of reusable objects in the Reuse Library:

- Feature template
- Law curve
- Punch, rib, shape, and snap

Where do I find it?

Resource bar	Reuse Library

Reusable object enhancements

These enhancements let you:

- Add a reusable component to a feature pattern.
- Select a template part for the reusable component if it is not found during editing.
- Drag a knowledge enabled part family member from the Teamcenter classification to the graphics window.

Where do I find it?

Resource bar	Reuse Library
Reuse Library Navigator	In the **Member Select** area, right-click a reusable component and choose **Insert**. Drag a reusable component from the **Member Select** area to the graphics window.

Fastener Assembly

Customize Fastener Assembly

Use the **Customize Fastener Assembly** command to add your own parts to the **Fastener Assembly** library. This command registers your parts as standard fastener assembly parts by updating the knowledge definition files. Knowledge definition files include the *HoleFastenerMap.krx* file and the *ReuseLibraryFasteners.krx* file that are defined by the customer default settings.

The registration process begins with a check for standard parts. A selected part is rejected if it is not a knowledge enabled part. You must do the following:

- Specify which standard part parameter is used for the diameter of the hole and the length of the hole.
- Add the selected part to the knowledge definition files.
- Specify the hole type for the parts in the knowledge definition files.

Where do I find it?

Toolbar	**Reuse Library® Customize Fastener Assembly**
Menu	**Tools® Reuse Library® Fastener Assembly Assistant® Customize Fastener Assembly**

Remove Fastener Assembly Node

Use the **Remove Fastener Assembly Node** command to remove the top node of a fastener assembly. This command finds all fastener assemblies in the assembly and removes the top nodes automatically.

Where do I find it?

Toolbar	**Reuse Library® Remove Fastener Assembly Node**
Menu	**Tools® Reuse Library® Fastener Assembly Assistant® Remove Fastener Assembly Node**

Fastener Assembly customer default enhancements

You can now use the **Customer Defaults** dialog box to set the following:

- Library path for the fastener assembly
- Path to configuration files in Teamcenter when you run NX in Teamcenter Integration mode.

Where do I find it?

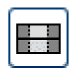
Menu	**File® Utilities® Customer Defaults® Gateway® Reuse Library ® Fastener Assembly**

Reusable Pocket enhancements

These enhancements let you:

- Break a pocket link from your component.
- Add a default extended length for threaded holes.
- Support add Material.

Where do I find it?

Toolbar	**Reuse Library® Reusable Pocket** 
Menu	**Tools® Reuse Library® Reusable Pocket**

Product Template Studio

Usability enhancements

What is it?

As the result of the usability enhancements, you can now add the following parameters to a template dialog box:

- Optimization study
- Point, vector and Boolean expressions
- NX requirement checks
- HD3D tags
- General relinker options
- Material properties

- Read only text boxes for string parameters

- A help tag for the template dialog box to behave like context sensitive help

Other enhancements include the ability to:

- Modify the text box size in the template dialog box. This is to support non-English template dialog boxes.

- Store and access template images in Teamcenter in bitmap format and use them for template labels.

- Store other data types like HTML, PDF, Excel spreadsheets, and Word documents used in templates in Teamcenter.

- Have the template dialog box updated automatically after you run a visual rule. The template dialog box displays updated parameter values when a visual rule changes the value of an expression.

- Select to have the **Redefine Constraint** dialog box appears when you drag a template part to the graphics window from the Reuse Library. Use this when your template part contains remembered constraints defined by the **Remember Assembly Constraints** command.

- Select to have the template component positioned at the drop point instead of the global origin when you drag it from the Reuse Library to the graphics window.

Where do I find it?

Application	Product Template Studio
Prerequisite	You must open a model or a template.

Visual Rule enhancement

What is it?

When you use the **Add Rule** command, you can now add a visual rule to:

- Launch a .NET application.

- Replace a component in the product template.

- Control the text for a label in the template dialog box. When a label no longer represents what the parameter controls as the result of a template operation, your visual rule can update the label to represent what the parameter controls.

Where do I find it?

Application	Product Template Studio
Prerequisite	You must open a model.
Toolbar	**Template Operations® Add Rule**

Initial values for template parameters

What is it?

When you add a product template to an assembly, NX scans the existing expressions in the assembly. If an expression has the same name as a parameter in the template dialog box, the value of this expression is used as the initial value of the parameter in the template dialog box.

Note When working with routing templates and assemblies, NX scans the expressions of stock components as well.

Where do I find it?

Prerequisite	You must add a product template to an assembly.

Add Routing Placement Solution Control

What is it?

Add the **Add Routing Placement Solution Control** dialog component to a routing template part to let the template user cycle through connection points and select the desired one. This dialog component appears in the template dialog box when you add the template part to an assembly.

Note This dialog component is similar to the **Placement Solutions** group available in the **Place Part** and the **Routing Solver** dialog boxes.

Where do I find it?

Application	Product Template Studio
Graphics Window	Dialog component bar

HD3D tags in product templates

What is it?

As the result of this enhancement, you can now add HD3D tags to a product template to do the following:

- Provide a link to documentation for the template.
- Provide usage guidance for the template.
- Highlight failures or warnings generated by requirement checks in the template.
- Document and guide WAVE re-parenting operations in the template.

Where do I find it?

Application	Product Template Studio
Prerequisite	You must add HD3D tags to a product template.

Routing Systems

Routing Systems

Part placement enhancements

What is it?

The **Instance Name Lookup** command is new.

You can now create a part instance name table in a Comma Separated Values (.csv) file format and use the new **Instance Name Lookup** command to place parts based on the instance name specified in the table.

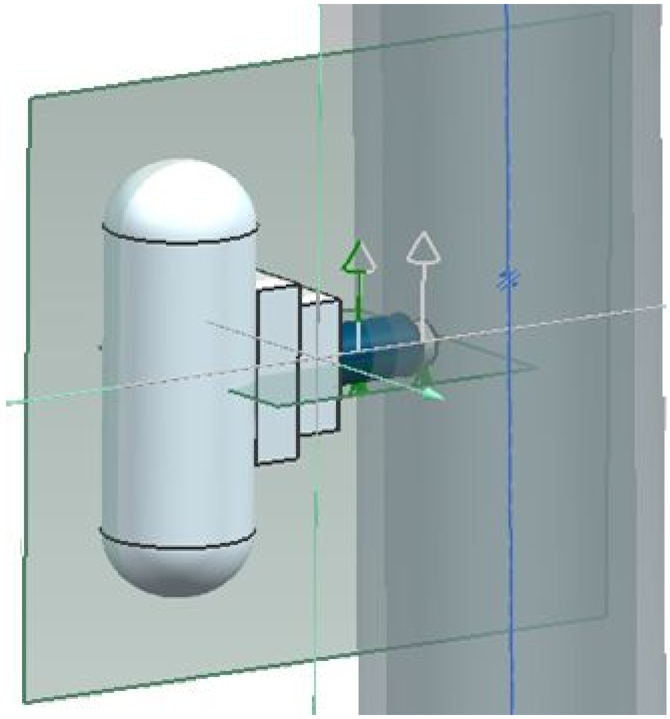
The **Place Part** command is enhanced.

You can now automatically select and place a part based on the destination characteristics and the run definition by using the **Place Part** command. The filter or the part **Search** criteria is based on the specification rules and relationships that you define.

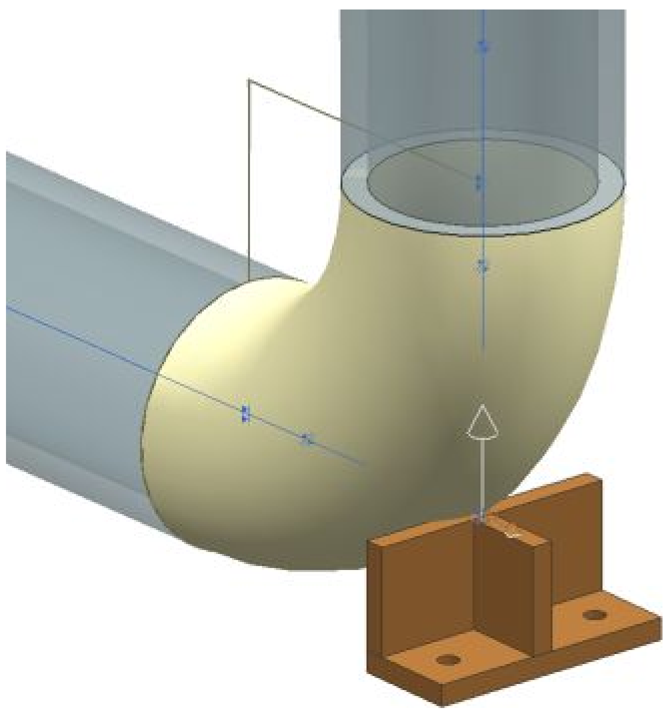
The parts are placed only if they do not disturb the flow or continuity of the pipes at the selected destination. For example, you cannot place an elbow at a pipe center as it disrupts continuity of the flow along the pipe.

You can also define the screw seat location for a part using the options in the new **Measurement Holder Location** group of the **Place Part** dialog box.

Place a mounting or screw seat under a pipe by selecting a point on the stock segment or an RCP. Specify the distance by which to offset the screw seat from the RCP. This offset is usually the radius of the stock or pipe.



Screw seat placed on a stock with an offset from the segment



Mounting placed at an anchor on an elbow joint

Why should I use it?

These enhancements help you to automate part selection and placement tasks and reduce modeling time. For example, you can define a rule to automatically place a flange every time you place a valve.

Where do I find it?

Instance Name Lookup command

Application	**Routing Mechanical**
Prerequisite	A part instance table must be defined and the ship number must be specified using the **Ship Identifier** customer default under **Routing**® **General** in the **File**® **Utilities**® **Customer Defaults** dialog box.
Menu	**Tools**® **Instance Name Lookup**

Place Part command

Application	Routing
Toolbar	Routing toolbar® **Part Drop-down** list® **Place Part**
Menu	**Insert**® **Routing Part**® **Place Part**

Create Fabrication enhancement

What is it?

You can now add additional fabrication attributes using the options in the new **Attributes** group and Optional subgroup in the **Create Fabrication** dialog box.

Why should I use it?

You can add more description to the fabrication without opening the *ugroute_mech_xxx.xml* file.

Where do I find it?

Application	Routing Mechanical
Toolbar	**Routing Mechanical**® **Part Drop-down** list® **Create Fabrication**
Menu	**Insert**® **Routing Part**® **Create Fabrication**
Location in dialog box	**Attributes** group® **Optional** subgroup

Design Rule Violation Objects Color

What is it?

The new **Design Rule Violation Objects Color** customer default allows you to change the color of an object that violates design rules.

In the case of a spline bend radius design rule violation, the specified color is applied to the temporary spline segments that show the violation of the bend radius, and not to the entire spline. In other cases, the color is applied to all the displayable objects that violate the design rule.

Why should I use it?

This customer default helps you to easily recognize design rule violations.

Where do I find it?

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Routing**® **General**® **Display** tab® **Design Rule Violation Objects Color**

Routing checks

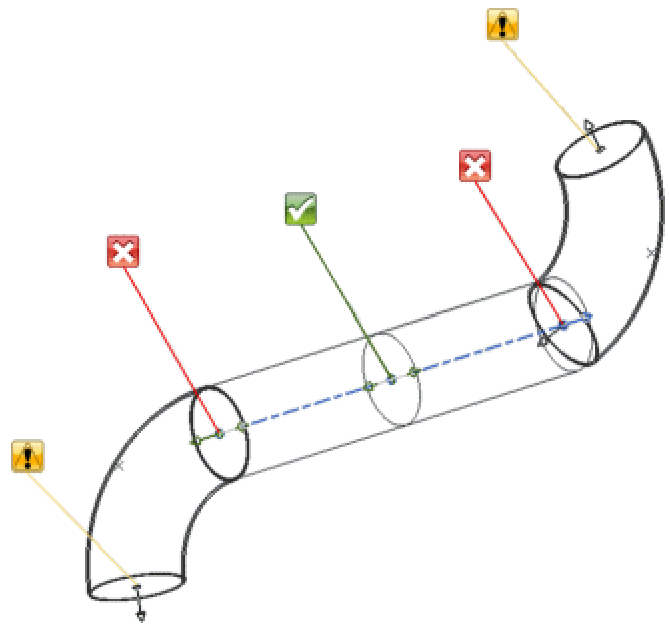
What is it?

You can now test the validity of Routing connections and display the results.

You can:

- Check parts in Routing Mechanical and Routing Electrical applications.
- Configure a callback to determine the validity of connections. If you do not use callbacks, NX relies on design rule results to check connection points in a part. To determine the validity of the connections, NX checks if the vectors for the connected ports are opposite and collinear.
- Override invalid connection points and record a justification. The overridden invalid connection points appear as valid in the display.
- Filter the results of the checks according to status of connection points. For example, you can view only invalid connections.

A visual display of the results of the check is created.



	Valid	Indicates that the connection point was determined to be valid by the tests that were run.
	Invalid	Indicates that the connection point was determined to be invalid by the tests that were run.
	No connection found	Indicates that the connection point contains a single port.
	Overridden	Indicates that an invalid connection point was changed to valid.

You can set up the tests in the following ways:

- Use Check-Mate tools.
- Launch the callback.

You can view the results in the following ways:

- Use Check-Mate tools.
- Use the HD3D visual reporting tools.

Why should I use it?

You can customize the callback and specify your own conditions and parameters for checking the connections.

Where do I find it?

The callback location is *ugroute_mech\plugins*.

Application	Routing
Toolbar	**Check-Mate® Set Up Tests** /**View Check-Mate Results**
Menu	**Analysis® Check-Mate® Set Up Tests/View Check-Mate Results**
Resource bar	**HD3D Tools** tab ® double-click **Check-Mate**.

Edit Line Segment enhancement

What is it?

The **Edit Line Segment** dialog box has a new **Detach Active Control Point** option. Use this option to delete all the constraints on the active RCP.

When you use this option, NX detaches the segment from all connections and creates new RCPs at all the detached ends. You can then use the new RCPs to move the line segment without affecting other line segments that were previously attached.

To reconnect the detached line segment, use the **Connect Path** command.

Where do I find it?

Application	Routing Logical and Routing Mechanical
Toolbar	Routing toolbar® **Edit Path Drop-down® Edit Line Segment**
Menu	**Edit® Routing Path® Edit Line Segment**
Graphics window	Right-click a route line® **Edit Line Segment**
Location in dialog box	**Settings** group® **Detach Active Control Point** button

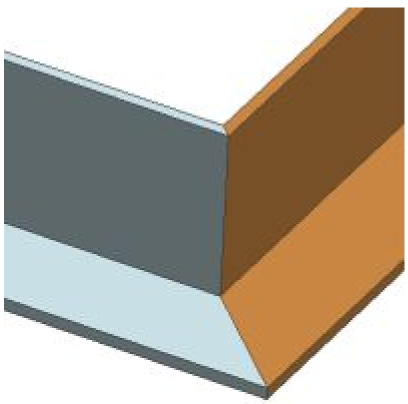
 **Assign Corner enhancements**

What is it?

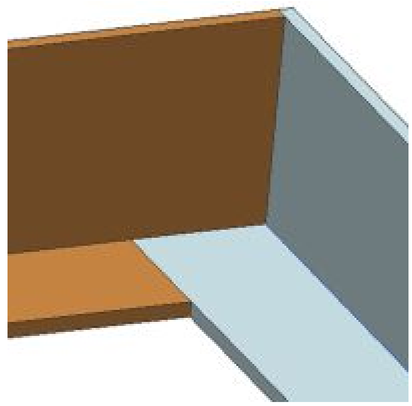
The **Assign Corner** command now includes the functionality of the **Cope Corner** command, which is no longer available.

When you use the **Assign Corner** command, you can now do the following:

- Create the following new types of corners:
  - **U-Bend**
  - **U-Elbow**
  - **Mitered Bend**
- Change the type of the corner from mitered to cope or cope to mitered.



Miter corner



Cope corner

Why should I use it?

Use the **U-Bend** corner type when you want to create a U-bend using two RCPs or three stock segments.

Use the **U-Elbow** corner type to assign elbows to RCPs when the side segments are parallel and it allows 0 mm gap between the two elbows, U-Elbow assigns cut-elbows when the two side segments are not parallel.

Use the **Mitered Bend** option to create a bend similar to bend with radius option, it allows user to specify number of miters in the bend.

Where do I find it?

Application	Routing
Prerequisite	The **Cope** command is available only in Routing Logical and Routing Mechanical.
Toolbar	Routing toolbar® **Path Drop-down** list® **Assign Corner** 
Menu	**Insert**® **Routing Path**® **Assign Corner**
Location in dialog box	**Corner** group® **Corner Type** list® **U-Bend** or **U Elbow** or **Cope** or **Mitered Bend**

Routing Mechanical

Run Navigator commands

What is it?

New shortcut commands are available from the **Run Navigator**.

Split Run

Splits a run into two separate runs.

You can change the default name for the split run by using the options provided in the **Run Name** group in the **Split Run** dialog box. The default name used for the split run is created by appending the name of the original run with a text string. You can use the **Split Run String** customer default to customize the text string.

Integrate Runs

Integrates two runs to create a unified run.

Reverse Flow

Reverses the direction of components and symbols that have characteristics defined for inlet and outlet ports, and that have collinear inlet and outlet points.

This command automatically interchanges the **From** and **To** items of a run.

Where do I find it?

Run Navigator commands

Application	Routing Mechanical and Routing Logical
Prerequisite	The **Integrate Runs** command is available only when you select two runs in the **Run Navigator**.
Menu	**Tools**® **Run Navigator**® **Split Run** or **Integrate Runs**
Run Navigator	Right-click a run® **Split Run** or **Reverse Flow** or **Integrate Runs**.

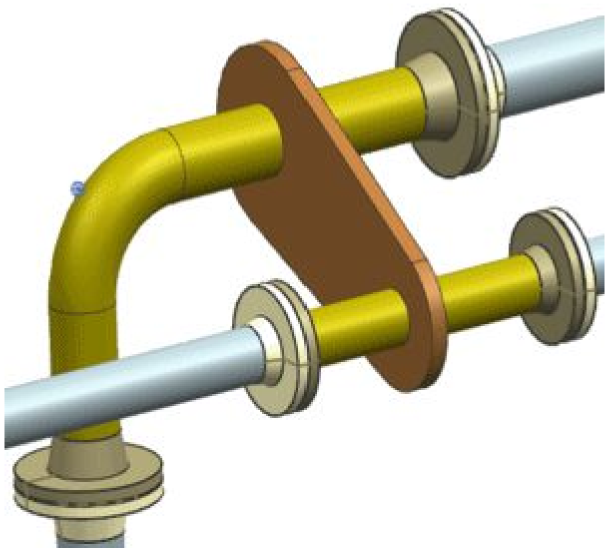
Split Run String customer default

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Routing**® **Mechanical**® **General** tab® **Run** group

 **Watertight Fittings**

Use this command to create watertight fittings and add them to a new subassembly. You can create watertight penetrations by creating additional hardware around a selected pipe or around multiple pipes that penetrate a wall.

The watertight fitting in the watertight subassembly shown was created from a sketch curve.



Where do I find it?

Application	**Routing Mechanical**
Toolbar	**Routing Mechanical**® **Tools Drop-down** list® **Watertight Fittings** 
Menu	**Tools**® **Watertight Fittings**

Routing Specification commands

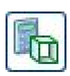
What is it?

Use the routing specifications command to create new routing specifications, edit existing specifications, and specify current specifications. In previous releases, project leads or managers had to create specifications by editing the APV files. The new specification commands help you to:

- Create specifications interactively, from a run, or from the P&ID diagrams.
- Search compatible parts and define part placement rules, automatically.
- Use the **Compatibility Tables** option to specify valid companion parts that are added when a connection is made.
- Use the **Generic Post Placement** option to specify an additional part that you want to be placed along with the part you place.

Where do I find it?

Application	**Routing Mechanical**
Prerequisite	You must define at least one discipline such as Tubing or Piping.
Reuse Library	Right-click the **Routing Specification** node® **Create New Specification**

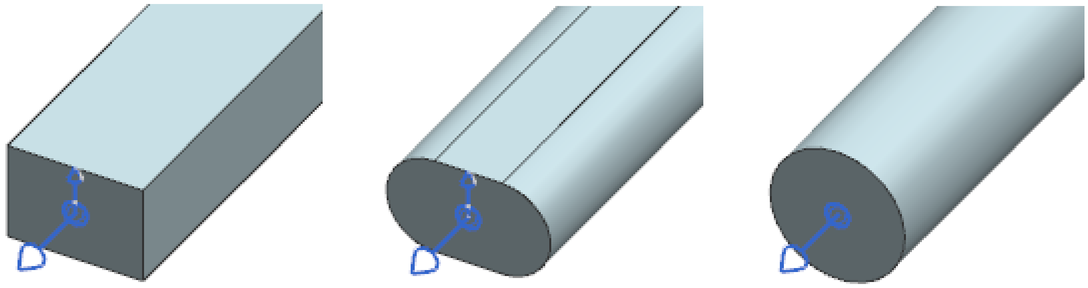


Duct Size Calculator

What is it?

Use this command to calculate the size of ducts used for HVAC applications. You can use this command to calculate parameters of ducts that have the following cross sections:

- **Rectangular**
- **Flat Oval**
- **Circular**



Rectangular **Flat Oval** **Circular**

Why should I use it?

You can:

- Calculate the size of the cross section of the duct based on the flow parameters.
- Adjust the flow parameters based on a specified duct size.
- Assign the calculated flow parameters to the selected segments as attributes of the segment, and display the parameters.
- Edit the flow parameters or cross-section parameters of multiple segments simultaneously if the parameters are identical.
- Create space reservations if required.

Where do I find it?

Application	Routing Mechanical
Toolbar	**Routing Mechanical**® **Tools Drop-down** list® **Duct Size Calculator**
Menu	**Tools**® **Duct Size Calculator**

Create Linear Path enhancements

What is it?

When you use the **Create Linear Path** command, you can now make eccentric pipe connections on a path. New options let you specify the direction and distance from the centerline of the initial path to define a branch that is eccentric.

Use the:

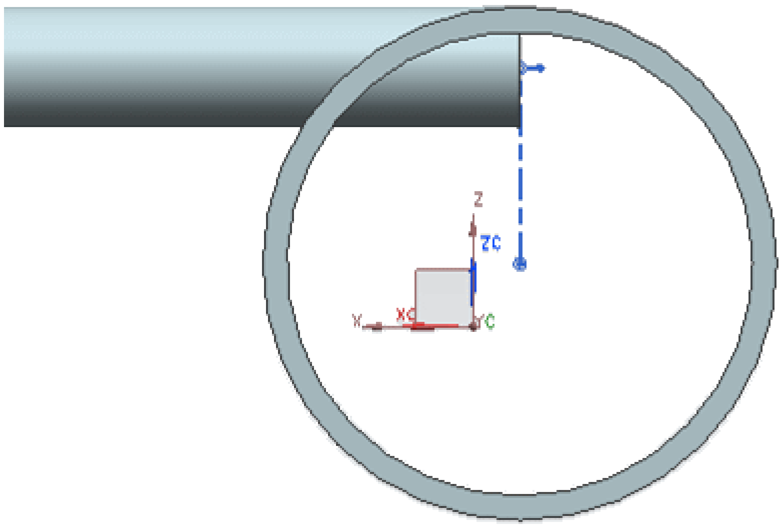
- **Eccentric Branch Point** type to create eccentric branch segments. You can select this option only after specifying a point on the curve from which you want the branch to begin.

- **Specify Vector** option to specify a direction for the eccentric branch.

  The default direction is down where down refers to the routing definition.

- **Eccentric Offset** option to specify the distance.

When you create the eccentric branch segment, the parent segment is subdivided at the specified point location.



An eccentric branch segment has the following characteristics:

- It is not associative to its parent segment and therefore its length is not updated when the stock on the parent segment changes.

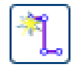
- You cannot assign stock to an eccentric branch segment.

- You cannot assign bend corners at the end RCP of an eccentric branch segment.

- You can change an eccentric branch segment to a normal segment and a normal segment to an eccentric branch segment. To do this, you must select the **Eccentric Branch Segment** check box in the **Edit Line Segment** dialog box.

Where do I find it?

Create Linear Path dialog box

Application	Routing Mechanical
Prerequisite	The eccentric branch options are available only after you specify a start point on a linear segment.
Toolbar	**Routing Mechanical® Create Linear Path** 
Menu	**Insert® Routing Path® Create Linear Path**
Location in dialog box	**Specify Point** group **Mode** list® **Eccentric Branch Point** **Eccentric Offset** **Match Eccentric Stock End** check box

Edit Line Segment dialog box

Application	Routing Mechanical
Toolbar	**Routing Mechanical® Edit Path Drop-down** list® **Edit Line Segment** 
Menu	**Edit® Routing Path® Edit Line Segment**
Location in dialog box	**Settings** group® **Eccentric Branch Segment** check box

Pipe End Prep plug-in

What is it?

You can use this plug-in to:

- Create a point at the connection with weld attributes on it.
- Create connection notes about end preparations, manufacturing instructions, and so on, using an attribute such as PIPE_END_NOTES on the pipe end port.

NX uses the plug-in when any two ports are connected to form a new connection.

Why should I use it?

The plug-in stores the pipe end preparation information that is required during the cutting or manufacturing process.

Where do I find it?

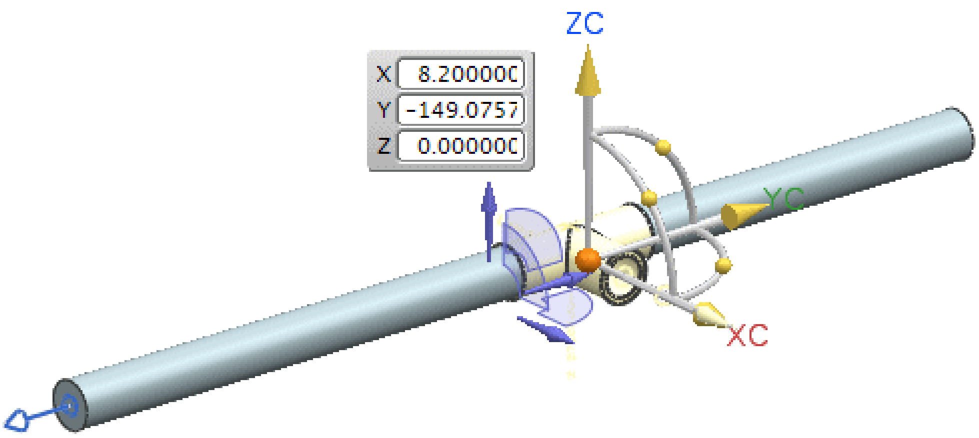
The plug-in location is *ugroute_mech\plugins*.

Transform Path enhancements

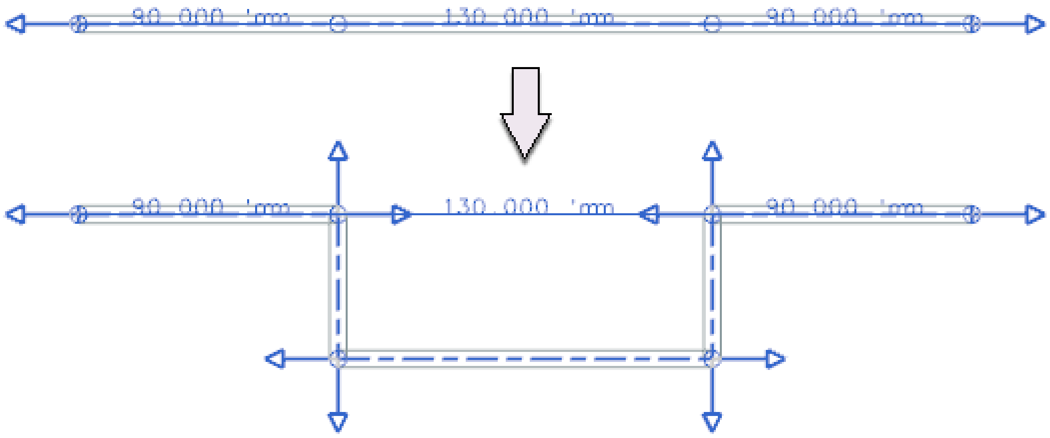
What is it?

Using the **Transform Path** command, you can now:

- Copy objects from one assembly to another assembly.
- Inherit specified attributes from the destination within the same assembly or across assemblies.
- Maintain constraints when you move or copy routing objects across assemblies.
- Automatically connect the routing objects at selected destination, if the destination is valid.
- Move a part using the **Dynamic** motion option.



- Use the **Copy Connected Parts** check box to automatically move or copy connected parts. It also subdivides a segment automatically when you copy another segment onto it.
- Use the **Move with Extension Segments** check box to easily create U-shaped paths.



Why should I use it?

The **Copy Attributes** group helps you move or copy routing objects with:

- Stationary attributes that you want to continue at the new destination.
- Destination attributes that you want to inherit from the selected destination.

The **Dynamic** motion enhancement helps you move a part along the axis or rotate around the axis.

Where do I find it?

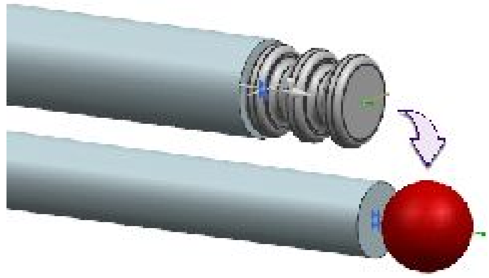
Application	**Routing Mechanical**
Menu	**Edit® Routing Path® Transform Path**
Toolbar	**Routing Mechanical® Edit Path** **Drop-down® Transform Path** 
Location in the dialog box	**Copy Attributes** group® **Defaults/User Specified** **Transform** group® **Motion** list® **Dynamic** **Settings** group® **Copy Connected Parts** check box **Settings** group® **Move with Extension Segments** check box

 **Unify Path enhancements**

What is it?

A new **Place Dummy Symbol for Parts not found** customer default is available. If this customer default is selected, you can

- Place dummy symbols in a routing path for the parts and stocks that are not available in the part library.



- Replace the symbols with parts and stocks that have matching characteristics after you update the library.

Why should I use it?

This enhancement helps you detect and visually mark parts and stocks that do not match specifications.

Where do I find it?

Application	Routing Mechanical
Customer Default	**Routing Mechanical** ® **General** tab® **Unify Path** group ® **Place Dummy Symbol for Parts not found**

Additional Pipe Length

What is it?

Use this command to add additional length to specified pipe ends.

You can use NX Open plug-ins to determine details such as:

- The minimum length that is necessary for a specific bending machine.
- The additional length that is necessary to account for weld shrinkage.

You can use this command on connected and unconnected ports. Connected ports are hidden by default. You can make the connected ports visible by using the new **Display Connected Ports** Routing preference.

For more information on visualizing templates, see **Unsatisfied xref title**.

For more information on checking and calculating pipe production length, see **Unsatisfied xref title**.

Where do I find it?

The callback location is *ugroute_mech\plugins*.

Additional Pipe Length command

Application	Routing Mechanical
Toolbar	**Pipe Production Length® Additional Pipe Length**
Menu	**Tools® Additional Pipe Length**

Display Connected Ports Routing preference

Application	Routing
Menu	**Preferences® Routing**
Location in dialog box	**Routing Preferences** dialog box® **Display** tab® **Display Connected Ports**

Visualize Templates

What is it?

Use the **Visualize Templates** command to display the locations at which pipe template lengths are applied.

You can:

- Display one or more of the following: ports, segments, stock.
- Display templates based on the template string, template values, or both.
- Display templates in only the work part or in all displayed parts.

For more information on creating additional pipe lengths, see **Unsatisfied xref title**.

For more information on checking and calculating pipe production length, see **Unsatisfied xref title**.

Where do I find it?

The callback location is *ugroute_mech\plugins*.

Application	Routing Mechanical
Toolbar	**Pipe Production Length ® Visualize Templates**
Menu	**Tools® Visualize Templates**

Check and Calculate Pipe Production Length

What is it?

Use this command to examine pipe ports received as inputs.

When you run this command, NX does the following:

- Determines which stock or pipe the port belongs to.

- Determined the appropriate bend machine to use.
- Read template length from the pipe end port and calculates the required cut length.

  If the template length is longer than the calculated cut length, the cut length is set to 0. In all other cases, the cut length is set equal to the difference, even if the difference is 0.
- Calculate the required shrinkage compensation.

  If the template length is longer than the calculated weld shrinkage, the weld shrinkage is set to 0.
- Locates the other stock port and computes the production length using the values from both end ports.

  The value of the production length is stored in the appropriate attribute on the pipe.
- Adds the following attributes as needed:
  - Adds attributes on the port to manage additional production length. For example, CUT_LENGTH, TEMPLATE_LENGTH, WELD_SHRINKAGE.
  - Adds an attribute on the stock to store the calculated production length.
  - Adds attributes on the stock to store the assigned bend machines.
- Check for other design rule violations, or cases where the additional production length cannot be calculated and reports the violations.

For more information on creating additional pipe lengths, see **Unsatisfied xref title**.

For more information on visualizing templates, see **Unsatisfied xref title**.

Why should I use it?

This command enables the cut lengths and weld shrinkages of a port to be updated based on changes made to the template length value.

Where do I find it?

The callback location is *ugroute_mech\plugins*.

The **Check and Calculate Pipe Production Length** command is run automatically when you click **OK** or **Apply** in the **Insert® Routing Stock® Additional Pipe Length** dialog box.

Application	Routing Mechanical
Toolbar	**Pipe Production Length® Check and Calculate Pipe Production Length**
Menu	**Tools® Check and Calculate Pipe Production Length**

Displaying routing stock as components in Teamcenter

What is it?

You can display routing fitting overstock and transition stock as separate components in Teamcenter when you select the **Stock as Components** routing preference.

If this preference is selected, you can also display the space reservation stock as separate components in Teamcenter, using the **Space Reservation as Components** option. The space reservation stock does not include non-routing space reservation such as light cones, sprinklers, or other modeling fixtures.

Why should I use it?

You can interact with these routing components in the Product Structure Editor without opening the part files in NX. You can also include these components in the Bill of Materials.

Where do I find it?

Stock as Components

Application	Routing Mechanical
Menu	**Preferences® Routing**
Location in the dialog box	**Stock** tab® **Stock as Components**

Space Reservation as Components

Application	Routing Mechanical
Prerequisite	The **Stock As Component** check box must be selected.
Menu	**Preferences® Routing**
Location in the dialog box	**Stock** tab® **Space Reservation as Components**

Converting legacy space reservation stock as components in NX

What is it?

You can now convert legacy space reservation stock to components in NX.

Why should I use it?

You can display the legacy space reservation stock as components in Teamcenter.

Where do I find it?

Application	Routing Mechanical
Menu	**Edit® Routing Stock® Convert to Stock as Components**

Comos Integration

What is it?

The Comos Integration connects Routing Mechanical with the Comos Process and Instrumentation Diagram (P&ID) tool. With this integration there are several useful tools now available in Routing Mechanical

Option	Description
Connect	Connects Routing Mechanical with the Comos application.
Disconnect	Breaks the connection between Routing Mechanical and the Comos application.
Unassign	Breaks the link between a symbol on the Comos P&ID and the component in your Routing Mechanical assembly.
Navigate To	Allows you to jump from a component in your Routing Mechanical assembly to the P&ID symbol or the 3D object in the Comos application.

From the Comos P&ID application there are new tools under the **NX Viper** menu:

Option	Description
Assign	Allows you to connect a symbol on the Comos P&ID to a component in your Routing Mechanical assembly.
Navigate	Allows you to jump from the Comos P&ID symbol to the assigned component in your Routing Mechanical assembly.
Route	Allows you to send the information for a route or run from Comos to Routing Mechanical . Routing Mechanical takes the information and creates or updates runs in the **Run Navigator**.

Why should I use it?

This enhancement creates a tighter coordination between your P&ID and 3D Routing Mechanical assemblies.

Where do I find it?

Application	Routing Mechanical
Menu	**Tools® Schematics**

Ship Design

Ship Structure applications

What is it?

The Ship Design application is broken into three new applications based on user tasks. The new applications are available when you use NX in a Teamcenter environment.

Ship Structure Basic Design

Use this application to design a high level representation of the ship structure.

For example, you can define structural systems such as decks and bulkheads. You can also divide the ship into zones based on strength, usage, and manufacturing requirements.

Ship Structure Detail Design

Use this application to design the detailed representation of the ship structure.

For example, you can create steel structure parts, apply detailed cuts to the ends of stiffeners, add chamfers between plates of different thicknesses, and add steel insulation.

Ship Structure Manufacturing

Use this application to model the manufacturing shape of the ship structure and augment that shape with manufacturing information.

For example, you can add material to plates or stiffeners, create rolling lines on plates to use in a forming process, and create marking lines to indicate the position of stiffeners for a welding process.

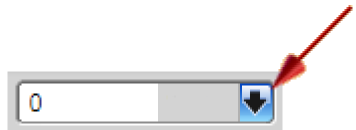
Note Some commands are available in multiple applications.

Concept Model enhancements

What is it?

The dialog boxes for the **Concept Model** commands are enhanced to improve clarity and to be consistent with other NX dialog boxes.

- You can access DesignLogic list options in numeric input boxes to specify a value based on a formula, a reference to an existing value, or a derived value from a measurement.



- Some dialog box options and groups are renamed.

- Some dialog box options are organized into new groups.

The **Concept Model** commands also include the following enhancements:

- In the **Main Dimensions** dialog box, you can define the **Midship** plane location. The default location is the middle of the **Aft Perpendicular** and **Forward Perpendicular** plane locations.

- In the **Main Dimensions** dialog box, you can specify the **Grid ID**. This option was moved from the **Export Frame Bar XML** dialog box and can be used in Drafting application to create frame bar objects.

- In the **Transverse Frame** dialog box, you can specify sets of frames in the –X direction.

- In the **Bulkheads** dialog box, you can create bulkheads along the Y-axis to reference longitudinal Y frames.

Where do I find it?

Application	Ship Structure Basic Design, Ship Structure Detail Design
Toolbar	**Ship Structure Basic Design**® **Concept Model Drop-down** list® All commands **Ship Structure Detail Design**® **Concept Model Drop-down** list® All commands
Menu	**Insert**® **Concept Model**® All commands

Ship Coordinates

Use the **Ship Coordinates** command to quickly obtain or specify the location of a point relative to ship grid reference planes, ship coordinates, and absolute coordinates without having to manually capture and convert coordinate values.

You can:

- Select a point or a port on a routing component and obtain the absolute coordinates and ship coordinates.
- Specify a location using absolute coordinates and obtain the ship grid coordinates.
- Specify a location using ship grid coordinates and obtain the absolute coordinates.
- Select reference planes along the X, Y, and Z axes of the ship grid.
- Create a point feature by specifying a location with absolute coordinates or ship grid coordinates.

Note The part file must contain existing ship grid planes created with the **Main Dimension** command.

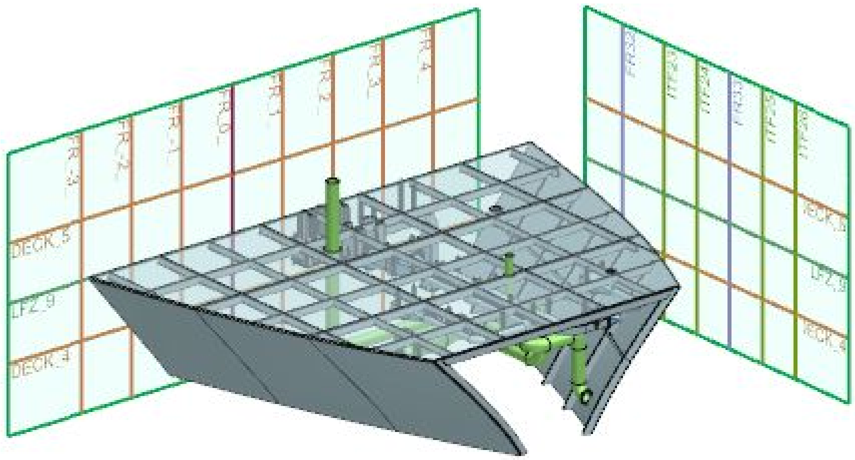
Where do I find it?

Application	Ship Structure Basic Design, Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Ship Coordinates**

Planar Ship Grid

Use the **Planar Ship Grid** command to create a planar set of selectable grid lines on a specified base plane. Each grid line represents a datum plane that intersects the base plane.

This command saves time by making it easier to select datum planes when you are placing components. If the datum plane you want to select is not located in the area you are working, you can select the corresponding grid line.



You can:

- Create grid lines for all or selected datum planes.
- Select a grid line instead of the corresponding datum plane.
- Specify line and color style for the grid lines.
- Display labels and specify the label color on the grid lines.
- Use filters to select, show, and hide the grid.

Where do I find it?

Application	Ship Structure Basic Design, Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Planar Ship Grid**

Ship Container

Use the **Ship Container** command to identify a parent assembly for new steel feature components. This ensures you add components in the correct hierarchy in a manufacturing block assembly without having to change the work part.

- A container part is required when you create the following types of features:
  - o Plates
  - o Stiffeners and edge reinforcements
  - o Pillars
  - o Steel insulation
  - o Standard parts

If there is no container part identified in the displayed assembly, the **Ship Container** dialog box is automatically displayed when you create the features.

- You can select any write-accessible part in the displayed part hierarchy as a container part except:
  - o Steel feature components.
  - o Components identified as piece parts using the **Enforced Piece Part** command.
- A SHIP_CONTAINER attribute is assigned to the container part so it can be identified in other NX sessions.

Where do I find it?

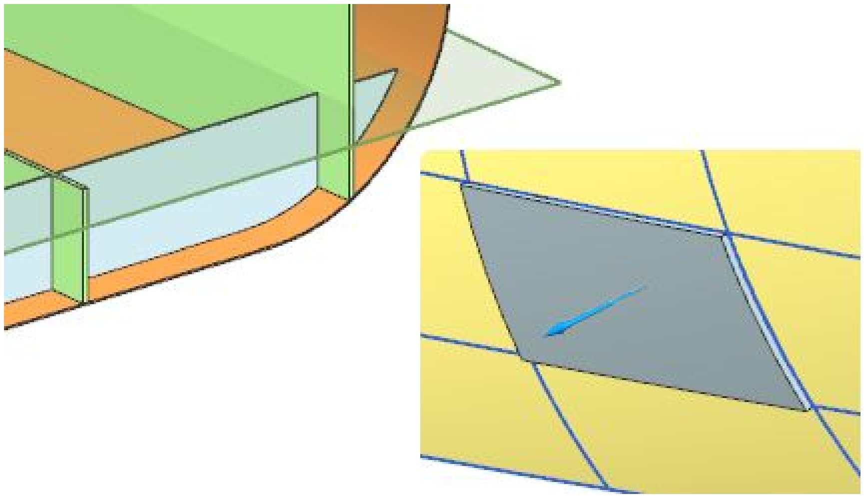
Application	Ship Structure Basic Design, Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Ship Container**

The dialog box may also be displayed automatically when you create steel features.

Application	Ship Structure Detail Design
Prerequisite	A container part has not already been identified among the displayed parts.
Toolbar	**Ship Structure Detail Design® Plate** or **Stiffener/Edge Reinforcement** or **Pillar** or **Steel Insulation** or **Add Standard Part**
Menu	**Insert® Steel Features® Plate** or **Stiffener/Edge Reinforcement** or **Pillar** or **Steel Insulation** or **Add Standard Part**

Plate

Use the **Plate** command to create structural plates and to carry the manufacturing information from boundary curves to plate edges during the design process. You can quickly create hull and bulkhead plates without having to use other modeling commands to trim and thicken sheet bodies.



You can:

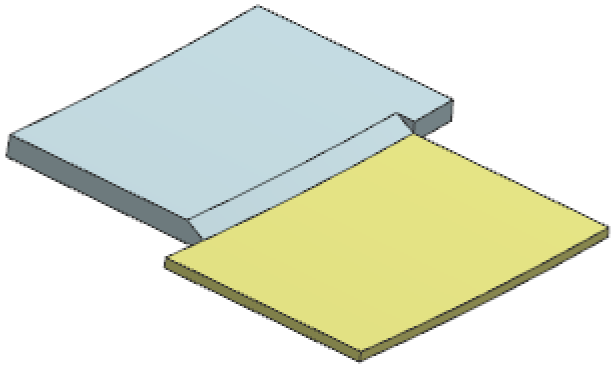
- Define the boundary of a plate by using construction curves such as butt and seam lines or by using faces and datum planes.
- Offset the plate from a selected face or datum plane.
- Create multiple plates at one time when inputs have multiple regions.
- Assign attributes to plates, edges, and bodies to identify common edges, molding line faces, opposing molding line faces, and calculate the mass, area, and volume of the solid body.
- Specify material and manufacturing information as attributes for process planning and weld specifications.
- Adopt an existing body as a plate.

Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Plate**
Menu	**Insert® Steel Features® Plate**

Plate Chamfer

Use the **Plate Chamfer** command to create a chamfer on a plate where it adjoins a thinner plate.



You can:

- Select plates from different components in an assembly.
- Associate the chamfer to the geometry and thicknesses of the adjacent plates.
- Specify start and end limiting planes.
- Chamfer both sides of a plate.

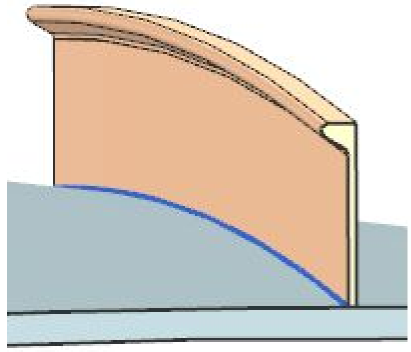
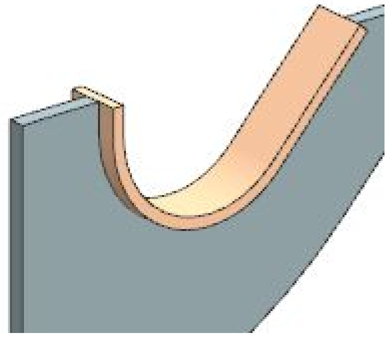
Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Plate Chamfer**
Menu	**Insert® Steel Features® Plate Chamfer**

Stiffener/Edge Reinforcement

What is it?

The **Profile/Plate** command is enhanced and replaced by a new **Stiffener/Edge Reinforcement** command. The new command has additional methods to place and limit geometry.

- The profile sketch library is now driven by spreadsheets.
- You can select datum planes to define guide curves. The feature is swept along the intersection of the selected datum planes and placement faces.
- When placing a profile, you can choose a predefined anchor point from the profile sketch. The anchor point will be aligned on the guide curve.
- The **Reverse Direction** option takes the inferred thickness into account. The feature is offset by the thickness so that it is located on the opposite face.
- You can define limits to trim or extend the feature by selecting geometry such as bodies, faces, or datum planes. Use the **Path Limits** options to control how the feature will be modeled when the limiting geometry is at an angle to the profile.
- You can orient stiffeners using the placement face normal, an orientation vector, a tilt angle, or multiple datum coordinate systems.
- You can select connected guide curves that are not G1 continuous to create knuckled profiles.

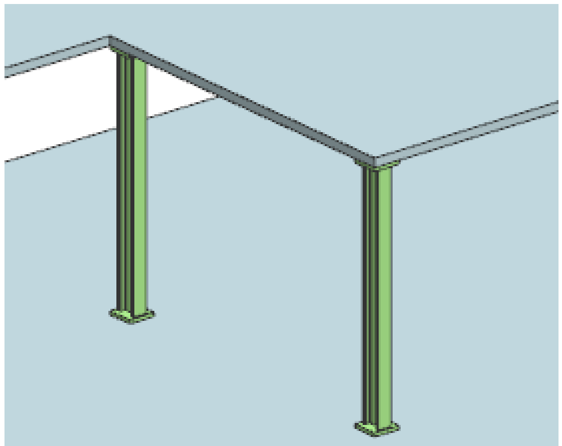
Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Stiffener/Edge Reinforcement**
Menu	**Insert® Steel Features® Stiffener/Edge Reinforcement**

Pillar

What is it?

The **Steel Support** command is enhanced and replaced by a new **Pillar** command.



- The dialog box layout and terminology are consistent with the other new steel feature commands.
- In the dialog box, the **Support Caps** group is renamed to **Pillar Caps**.
- The section sketch library and pillar cap sketch library are now driven by spreadsheets.

Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Pillar**
Menu	**Insert® Steel Features® Pillar**

Copy Parts between Planes

Use the **Copy Parts between Planes** command to copy multiple component parts by selecting a reference and destination object.

This is useful for copying existing plates and stiffeners to a different plane or mold face.

- The new components are added to the current work part.
- In the new components, geometry associated to the selected reference object, such as WAVE links, are re-parented to the destination object.

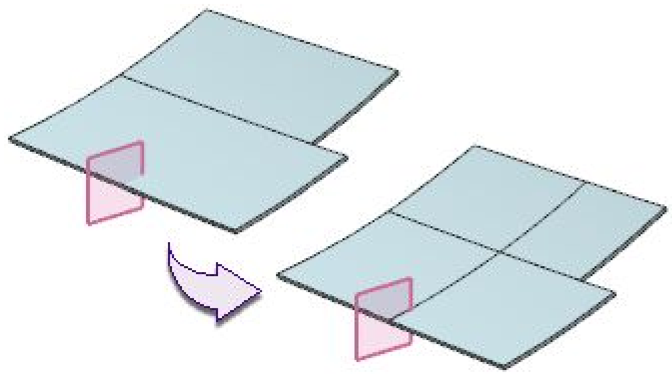
- The WAVE links not associated with the reference object remain linked to the original parent geometry.
- Attributes are maintained in the new component parts.

Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Copy Parts between Planes**
Menu	**Insert® Steel Features® Copy Parts between Planes**

Split Profile/Plate

Use the **Split Profile/Plate** command to split existing profile and plate parts into more manageable pieces. You can also split parts when adjacent structural parts have different thicknesses.



You can:

- Define the split location using a new or existing datum plane, faces, or by projecting curves to the split body in a specified direction.
- Select multiple profiles and plates to split at one time.
- Automatically select attached profiles on the plate to split at one time.
- Split profiles and plates using a direction that is perpendicular to the plate mold face, or orthogonal to both the profile base and mold face.
- Use the **Remove Split Feature** command to remove a split feature and the associated parts. The profiles and plates revert to their original size.

Where do I find it?

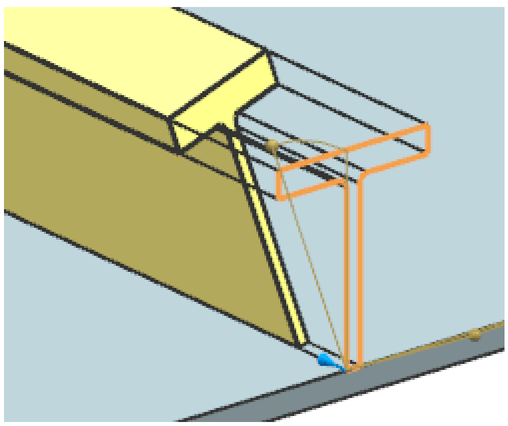
Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Split Profile/Plate**
Menu	**Insert® Steel Features® Split Profile/Plate**

End Cut enhancements

What is it?

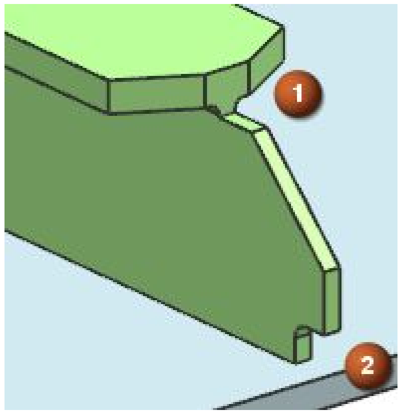
The **End Cut** command is enhanced so you can:

- Use the new **Inherit** option to copy the parameters from an existing end cut and apply them to one or more end cuts.
- Select multiple end faces.
- Create end cuts that are driven by flange, web, and bevel parameters.
- Tilt, shear, and offset the end of a profile before the end cut parameters are applied. The new **Shear Angle** box replaces the **Taper** option in the **Type** list under **Draft**.



- Apply the shoulder and toe cuts for the web with a single sketch or with two separate sketches. The sketch library is now driven by spreadsheets.

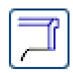
  In this example, separate sketches are used for the shoulder (1) and toe (2).



Why should I use it?

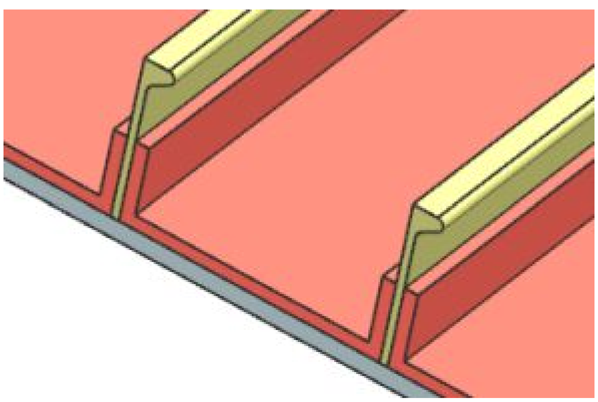
These enhancements save time by allowing you to quickly copy parameters of an existing end cut to multiple end cuts. You can also create end cuts driven by flange, web and bevel parameters, and tilt, shear and offset the end of a profile before the end cut parameters are applied.

Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® End Cut** 
Menu	**Insert® Steel Features® End Cut**

Steel Insulation

Use the **Steel Insulation** command to define, model, and evaluate insulation in a ship design for fire proofing, noise reduction, and heat retention. This command creates a feature that contains one or more solid bodies.



You can:

- Apply the insulation to one or more sets of boundaries on selected faces.
- Choose the type of insulation from a list of materials with thickness, density, colors, and attributes.
- Analyze the mass and area properties of the insulation.
- Unform the insulation to create a template for manufacturing.

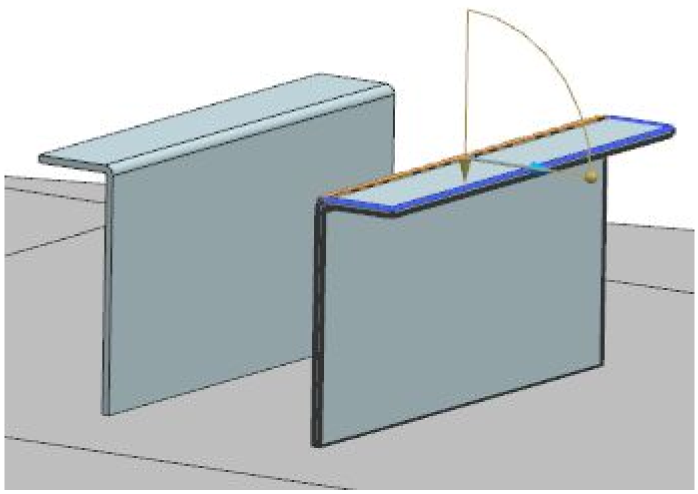
Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Steel Insulation**
Menu	**Insert® Steel Features® Steel Insulation**

Flange enhancements

What is it?

You can create a sheet metal Flange feature on a Plate feature that was created in the Ship Structure Detail Design application.



- You can access the **Flange** command in the Ship Structure Detail Design application.
- You can create a table to allow multiple thicknesses for each material and multiple bend radii for each thickness. These standards apply only to flanges created in the Ship Structure Detail Design application.

- Before adding a flange, you must convert the plate feature to a sheet metal feature. You can use the **Convert to Sheet Metal** command which is now available in the Ship Structure Detail Design application.

- You can specify a neutral factor for each material in the table.

Why should I use it?

Use this command to add flanges to plates without changing applications. You can also specify standard material thicknesses and bend radii that apply only to flanges created in the Ship Structure Detail Design application.

Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Flange**  **Ship Structure Detail Design® Convert to Sheet Metal** 
Menu	**Insert® Steel Features® Flange** **Insert® Steel Features® Convert to Sheet Metal**

Standard parts

What is it?

You can use the standard parts capabilities to define and add standard ship structural parts based on intelligent rules with minimal interaction.

You can:

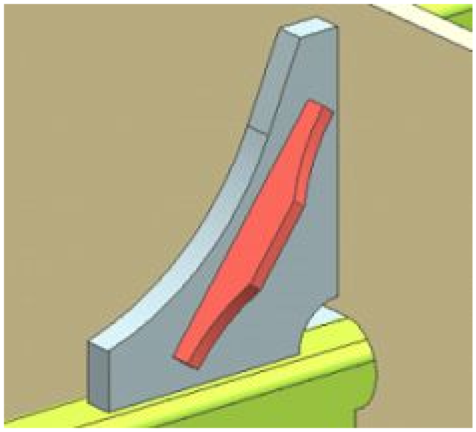
- Add different standard parts with a consistent user interface.

  Note NX provides standard part templates for tripping brackets, stiffener end brackets, and collar plates. The **Bracket** and **Collar Plate** commands are removed and replaced by categories in the **Add Standard Part** command.

- Select reference geometry to define the location and orientation of a standard part.

- Use commands to quickly edit and delete standard parts.

- Add standard parts with embedded design knowledge for your site. For example, a tripping bracket can have a rule to include a stiffener if the length exceeds a specified value.



If you create and maintain standard part templates, you can:

- Specify new categories and types of standard parts in a spreadsheet.
- Specify the parameters of a standard part in a spreadsheet.
- Define the configuration for the user interaction and attribute mapping of the standard part in an XML file.
- Define knowledge rules to provide intelligence for the parameters, location, and orientation of the standard part in an XML file.
- Specify the location of the spreadsheet, configuration file, and knowledge rules file using the **Standard Part Framework** customer defaults.

Where do I find it?

Adding standard parts

Application	Ship Structure Basic Design, Ship Structure Detail Design
Toolbar	**Ship Structure Basic Design® Add Standard Part** **Ship Structure Detail Design® Add Standard Part**
Menu	**Insert® Steel Features® Add Standard Parts**

Editing and deleting standard parts

Application	Ship Structure Basic Design, Ship Structure Detail Design
Prerequisite	Select a standard part component.
Menu	**Edit® Steel Features® Edit Standard Part** or **Delete Standard Part**
Graphics window	Right-click standard part component and choose **Edit Standard Part** or **Delete Standard Part**.
Assembly Navigator	Right-click standard part and choose **Edit Standard Part Component** or **Delete Standard Part Component**.

Specifying standard part spreadsheets and configuration files

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Ship Design® Standard Part Framework**

Spreadsheets driving steel feature libraries

What is it?

You can incorporate your standard steel feature tables in spreadsheets to drive parametric sketches in NX steel feature libraries.

You can now configure libraries for the following types of features:

- Stiffeners
- Edge reinforcements
- End cuts
- Pillars
- Profile cutouts
- Cutouts

You use several different types of files to implement a steel feature library:

Registration spreadsheet	Coordinates the libraries and specifies the locations of the parameter and model files for each section type.
Parameter spreadsheet	Configures the parameters and attributes of a section type and the name of the associated legend images.
Model file	A part file containing a single, parametric sketch for a section type.

Legend image	Displays the shape and key dimensions of the section type.

You choose the available section types and sizes in the dialog boxes of the appropriate steel feature commands.

Why should I use it?

Using a standard spreadsheet reduces the set up time and effort required to create a parametric sketch for each family of steel features. When you create a steel feature, performance is improved because NX only needs to load a single sketch.

Where do I find it?

To specify or change the location of the registration spreadsheet:

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Ship Design® Steel Features® Qualify Sketch** tab You specify the names and locations of the other supporting files in the registration spreadsheet.

To access a library when creating the steel features:

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Stiffener/Edge Reinforcement** or **Pillar** or **Profile Cutout** or **End Cut** or **Cutout**
Menu	**Insert® Steel Features® Stiffener/Edge Reinforcement** or **Pillar** or **Profile Cutout** or **End Cut** or **Cutout**
Location in dialog box	**Stiffener/Edge Reinforcement® Stock** group **Pillar® Section** group and **Pillar Caps** group **Profile Cutout® Cutout** group **End Cut® End Cut Options** group **Cutout® Sketch** group

Qualify Sketch

Use the **Qualify Sketch** command to help build, maintain, and validate sketch libraries used to create steel structures.

You can use this tool to:

- Open template parts and spreadsheets that define the sketch libraries.
- Validate a sketch to make sure it follows all design rules.
  - Verify the sketch defines a valid cross section.
  - Verify required attributes are assigned to the appropriate geometry.
  - Verify attributes have valid values.
- Identify reference, variable, and fixed dimensions.
- Select sketch curves required for placing the steel feature in downstream operations such as marking line locations.
- Add anchor points to define placement locations for sketches.

Why should I use it?

Use this command to make sure that the rules required for the input data of steel features are followed.

Where do I find it?

Application	Ship Structure Basic Design, Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Qualify Sketch**

Custom attributes for steel features

What is it?

Use a custom attribute catalog to define standard attributes for a site or a project and add them to steel features.

You can:

- Define a set of attributes for each type of steel feature. The attribute catalog is an XML file you edit outside of NX and specify in the customer defaults.
- Specify default values for attributes.
- Make attributes required or optional.

- Assign attributes automatically without additional interaction or display the **Attributes** dialog box when you create or edit steel features. Use the dialog box to edit attribute values or delete optional attributes.

Why should I use it?

Implement a standard set of attributes to ensure consistency for an enterprise, a site, or a project. The attributes associated with steel features can be used in downstream applications.

Where do I find it?

Adding standard attributes to steel features.

Application	Ship Structure Detail Design
Prerequisite	Define the attribute catalog and attributes for a steel feature.
Dialog box	If the options to show the **Attribute** dialog box are selected in the customer defaults, click **OK** or **Apply** after you edit or create a steel feature.

Specifying the attribute catalog and controlling the display of the **Attributes** dialog box.

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Ship Design® Project Settings® Customizable Attributes** tab

Cutting Side Face enhancements

What is it?

To improve functionality and performance, the following enhancements to the **Cutting Side Face** command are available.

- A new **Automatic** option allows NX to automatically detect and select all of the cutting side faces for the objects in the current study or assembly. Appropriate attributes are then assigned to the solids.

  Note The option to manually select cutting side faces is still available.

- The ability to flip the assignment of the cutting side attributes to the opposite face.

- Additional options for more precise control over the automatic selection of tangential and coplanar faces.

Where do I find it?

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Cutting Side Face**
Menu	**Insert® Manufacturing® Cutting Side**

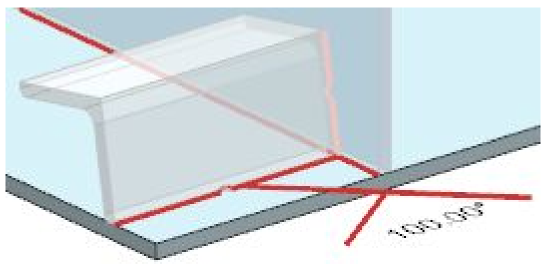
Marking Line enhancements

What is it?

The **Marking Line** command is improved with the following enhancements:

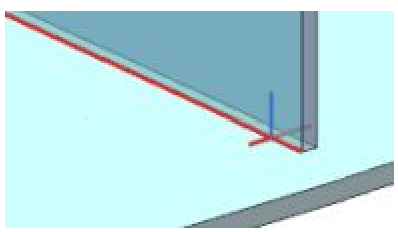
- You can create attachment marks for complex end attachments and canted profiles.

- Attachment marks for canted profiles on planar placement faces include angle template marks.



- There are four different attachment indicator types: **Bubble Shape**, **Bridge Shape**, **Triangle Shape**, and **End Bracket**. You choose the attachment indicator type in **Customer Defaults**.

- You can create match mark lines to show how two parts are aligned along the attachment mark. You specify the **Match Mark** parameters in the **Customer Defaults**.

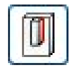


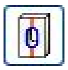
- You can select ship grid planes to use for reference lines.

Why should I use it?

Use this command to produce marking lines accurately and consistently and provide information to drive downstream processes for drawing automation and manufacturing output.

Where do I find it?

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Marking Line** 
Menu	**Insert® Manufacturing® Marking Line**



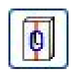
Reference Line enhancements

What is it?

To improve functionality and performance, the following enhancements to reference line features are made.

- The **Reference Line** command now creates core NX curve feature objects.
- The reference line object can be a single curve feature or multiple curve features.

Where do I find it?

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Reference Line** 
Menu	**Insert® Manufacturing® Reference Line**

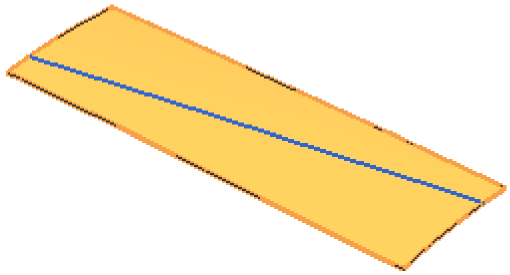
Rolling Line enhancements

What is it?

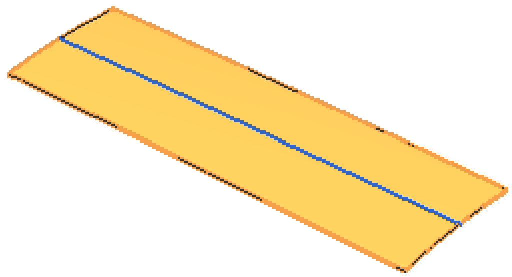
To improve functionality and performance, the **Rolling Line** command generates a core NX feature for each rolling line, using a standardized NX dialog box.

New options for the **Rolling Line** command let you do the following.

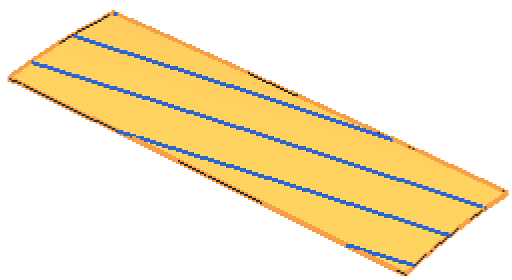
- Specify the type of rolling line to create. You can create the following types of lines.
  - A single manual line using a specified construction method. Available methods let you create a line derived from the surface curvature, or from an intersecting plane.



  - A geodesic line derived from a specified point and flow direction.



  - A series of pressure lines separated by an angular deviation.



- Specify a planar radius limit that lets NX ignore roughly planar regions.
- Use an analysis option to temporarily display a geometric and numerical representation of the surface curvature at a specified point on the surface face.

Why should I use it?

Use the different rolling line options to create more precise rolling lines.

Where do I find it?

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Rolling Line**
Menu	**Insert® Manufacturing® Rolling Line**

Plate Preparation enhancements

What is it?

The following enhancements are provided to improve your productivity when performing plate preparations.

- The **Plate Preparation** command handles generic formed surfaces in addition to straight break sheet metal parts.
- All plates are unformed using the Metaform tool.

Where do I find it?

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Plate Preparation**
Menu	**Insert® Manufacturing® Plate Preparation**

Template enhancements

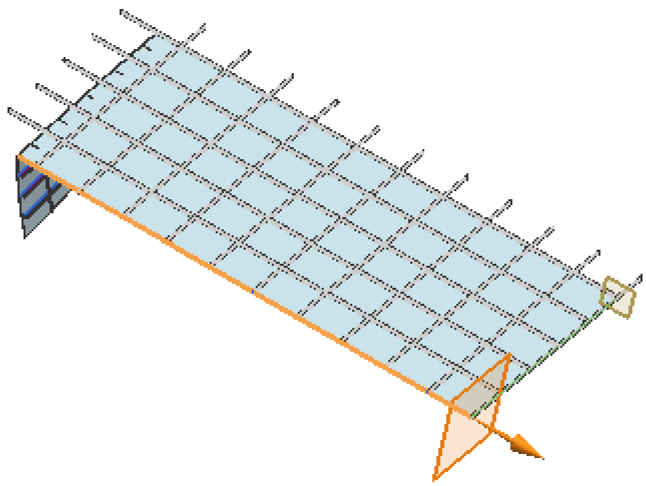
What is it?

To improve functionality and performance, the following enhancements to template features are made.

- The **Template** command creates an NX feature of editable sheet bodies using a standardized NX dialog box.
- A new **Infer Coordinate System** option allows NX to derive the template coordinate system based on selected surfaces and specified planes.

  Note The option to manually specify the template coordinate system is still available.

- New options to generate offset datum planes used to create the template sheet bodies.
- New options for specifying an optional thickness for the sheet bodies. Thickness is applied evenly to both sides of each sheet body.



Where do I find it?

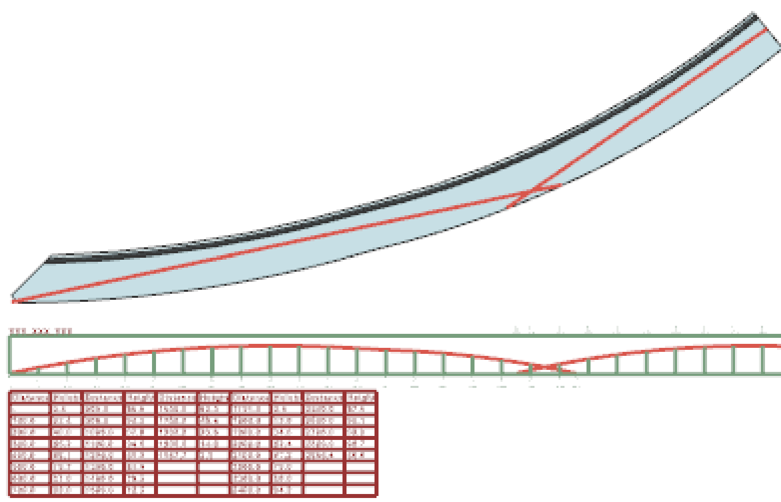
Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Template** 
Menu	**Insert® Manufacturing® Template**

Inverse Bending Lines enhancements

What is it?

The **Inverse Bending Lines** command is redesigned and improved with the following enhancements:

- An inverse bending line feature is created, and is associated to the defining profile. When the profile changes the bending line feature, which includes the bending lines, diagrams and tables, is also updated.



- The dialog box is redesigned and provides you with immediate control over more of the parameters used to calculate the bending lines.

  NX automatically detects the number of bent profiles in your assembly or study, and then uses dialog box settings to calculate the inverse bending lines, diagrams, and tables, for all the bent profiles.

- You can create inverse bending lines for twisted profiles.

- Bending line representations are more accurate.

- You can restrict users from manually changing the parameters used to calculate the bending lines and drawing data. You can set all parameters in the Ship Design customer defaults.

- You can create custom application programs using the Application Programming Interface (API) for C++, Java, and .NET.

Where do I find it?

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Inverse Bending Lines** 
Menu	**Insert® Manufacturing® Inverse Bending Lines**

Manufacturing view for ship design

Manufacturing view tools

You can use a set of manufacturing tools to place one or more 3D ship design elements into a manufacturing setting, and add manufacturing changes while keeping the design data intact. When you are in the manufacturing mode you can create, edit, and delete manufacturing modifications as needed for assembling and installing the design elements.

The manufacturing tools are comprised of a group of commands that let you do the following.

- Place a 3D design element into either a design mode or a manufacturing mode so that only those features applicable to the current mode are displayed.

- Identify and tag feature data, known as Metal To Metal features, which are used to modify a steel element, such as a stiffener or plate, so that no material gaps or material collisions exist between parts. These features are suppressed in the manufacturing mode.

- Compare the model differences between the manufacturing definition of one or more elements and the design definition of those elements.

- Synchronize the options and dimensions of a manufactured end cut to the original design end cut.

- List the design elements in a design study that contain design data, manufacturing data, or those which have not yet been through the manufacturing mode process.

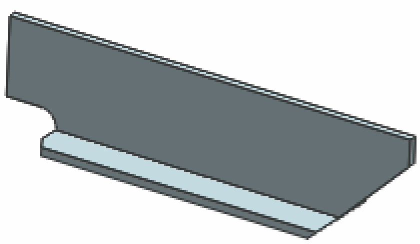
Where do I find it?

Application	Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Manufacturing**

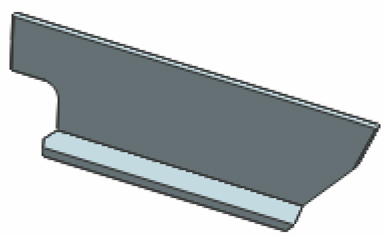
Manufacturing Data

Use **Manufacturing Data** to create a manufacturing version of one or more design elements. While in the manufacturing mode, all subsequent model changes are saved as manufacturing only data.

When the design element is in a manufacturing mode, you can add, delete or modify manufacturing features and details on the 3D design. The data you create when you are in the manufacturing mode is used to produce manufacturing information such as manufacturing drawings, assembly plans and flat patterns.



Stiffener in design mode



Stiffener in manufacturing mode

The changes you make when you are in the manufacturing mode are saved and stored with each element. In the manufacturing mode, the manufacturing data for the element is visible, and in the design mode, only the design data is visible.

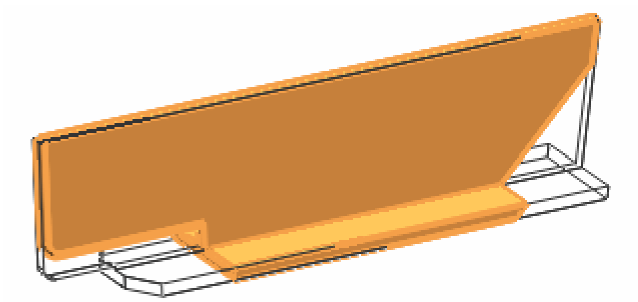
Where do I find it?

Application	Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Manufacturing® Manufacturing Data**

Compare

The **Compare** command lets you display both the manufacturing definition and the design definition for one or more design elements in a study or assembly so you can compare them. When you compare the two definitions, you can choose to emphasize either the design mode or the manufacturing mode. The emphasized mode is displayed as a solid body, and the alternate mode is displayed as a wireframe body.

In this example, the manufacturing mode is displayed as a solid body and the design mode is displayed as a wireframe body.



Note The **Translucency** option on the **Visual** tab of the **Visualization Preferences** dialog box must be set in order to see the wireframe body.

When you access another command, the comparison display is removed.

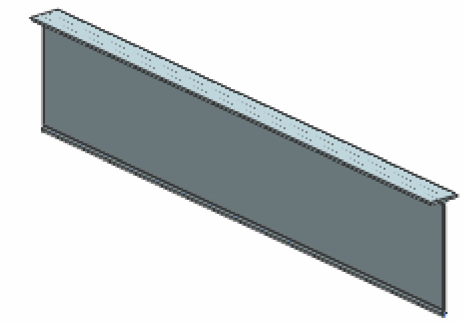
You can use the **Compare** command in a study that contains multiple design elements, or directly in the design element.

Where do I find it?

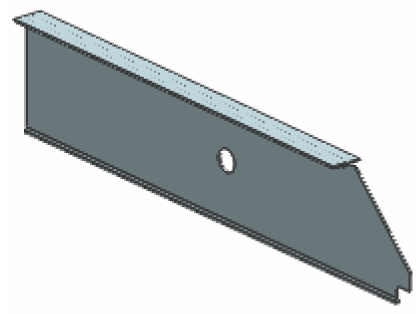
Application	Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Manufacturing® Compare**

Set Mode

Use **Set Mode** to change the state of one or more design elements. When you change the state of a design element, any subsequent changes you make to that element are placed in the current feature group (Design or Manufacturing).



Design mode



Manufacturing mode

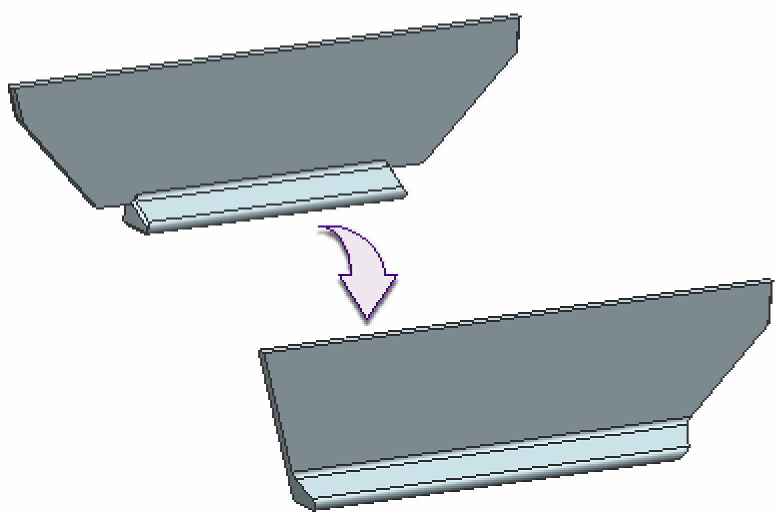
If you want features to show up in both modes, you must make sure you reorder the feature so that it occurs before the manufacturing group of features was created.

Where do I find it?

Application	Ship Structure Detail Design, Ship Structure Manufacturing
Prerequisite	The design element must have both a Design and Manufacturing feature group. These groups are created when the **Manufacturing Data** command is used.
Menu	**Tools® Ship Design® Manufacturing® Set Mode**

Rebase

Use the **Rebase** command to reset all the manufacturing parameters of an end cut back to the parameters of the design end cut. This command is useful when changes are made in the end cut of the original design after the manufacturing end cut is created.



You can use the **Rebase** command in a specific design element, or in a study or assembly if you want to rebase multiple design elements.

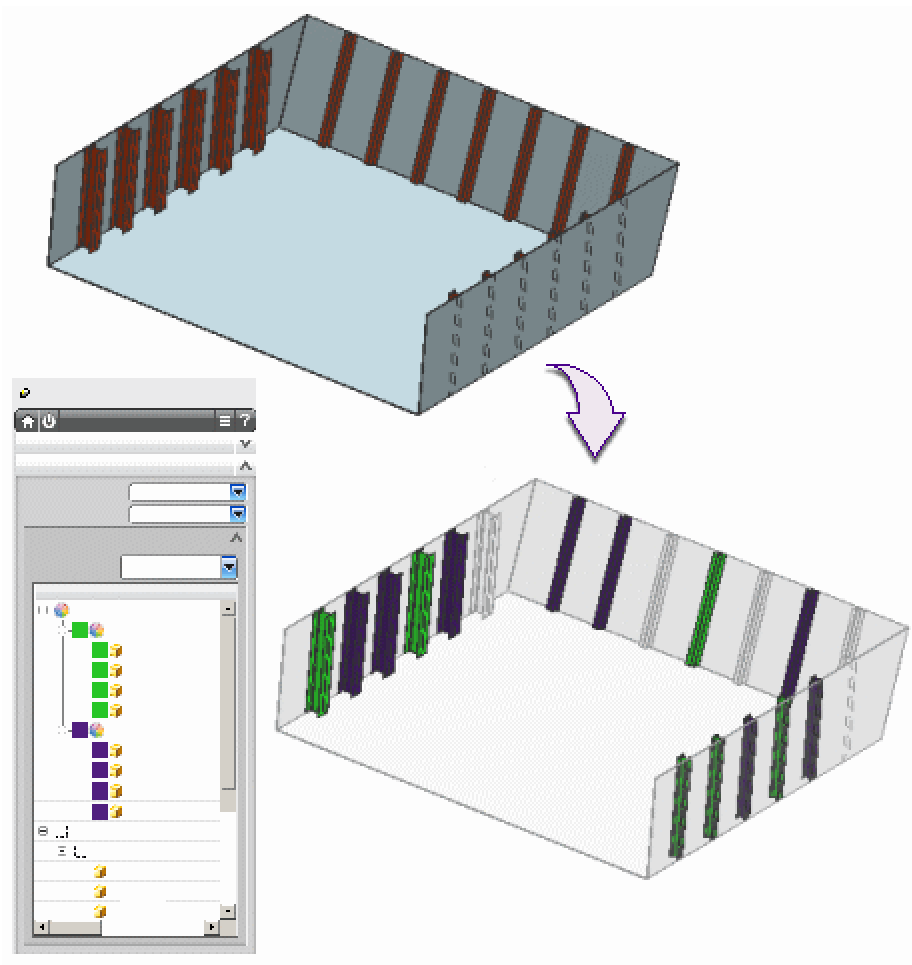
Where do I find it?

Application	Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Manufacturing® Rebase**

List

Use the **List** command to create a visual report that helps you identify the elements in your study or assembly that are in a manufacturing mode, those that are in a design mode, and those that have not been through the manufacturing mode process yet.

The data is displayed in the Resource bar in the **HD3D Tools** tab.



Where do I find it?

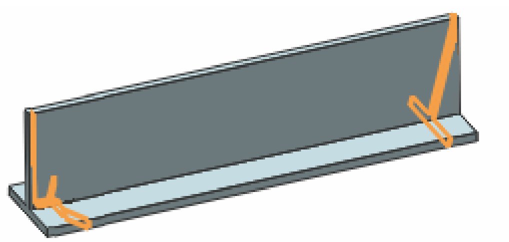
Application	Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Manufacturing® List**

Tag Metal to Metal Features

Use the **Tag Metal to Metal Features** command to identify features added to a steel element, such as a stiffener or plate, so that no gaps or material overlap between the steel element and its adjacent object are shown. These added features are suppressed when the steel element is placed in the manufacturing mode.

Note You must tag any feature on a steel element that you do not want to appear in the manufacturing mode first, before you place the element in the manufacturing mode.

In this example, a stiffener has added features which obscure the end cuts (end cuts are highlighted)



After the features are tagged, NX suppresses the tagged faces and displays only the original steel element when it is placed in the manufacturing mode. This allows the manufacturing engineer to alter the parameters of the original element, or replace it completely with a manufacturing feature.

Where do I find it?

Application	Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools® Ship Design® Manufacturing® Tag Metal to Metal Features**

Ship Drafting

Shipbuilding standards for Drafting

What is it?

A new shipbuilding drafting standards file is available that lets you automatically set drafting and annotation preferences and customer defaults in accordance with shipbuilding requirements.

Like other drafting standards files, you can customize the shipbuilding standards file to reflect company-specific drafting preferences.

Why should I use it?

Use the shipbuilding standard file to automatically set all your drafting and annotation preferences and defaults for your NX session.

Where do I find it?

To set the standard for the current NX session

Application	Drafting and PMI
Menu	**Tools**® **Drafting Standard**
Location in dialog box	**Standard to Load** group® **Standard** list® **Shipbuilding(Shipped)**

To set the shipbuilding standard as the default standard, or to customize the standard

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Drafting**® **General**® **Standard** tab® **Drafting Standard** list® **Shipbuilding(Shipped)**

Frame Bar enhancements

What is it?

The following enhancements are provided to improve and optimize your interaction with the frame bar functionality.

- The frame bar is now generated from core NX data derived from Concept Model information, rather than from an external XML file. You no longer need to set environment variables or create the XML file.

- **Horizontal** and **Vertical** frame bar options, available on the **View Style** dialog box, let you automatically create one or both frame bars while creating your view.

  Note These options are only available when you create the view. They are not available when you edit a view. To create a frame bar on an existing view, you must use the **Frame Bar** command. Note that you can only add frame bars to views with orientations that are parallel to one of the primary X, Y, or Z ship axes.

- Frame bars are specifically linked to the drawing view for which they are created. If you delete or move the view, the frame bar will be deleted or moved also.

- The frame bar is displayed using standard NX drafting objects, so normal NX commands can be used to edit the frame bar display.

- The frame bar is a single collection of objects. You cannot delete individual components of the frame bar.

- You can control the visibility, color, font and width of the different frame bar components. You can also set preferences for the default display of a frame bar.

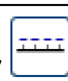
- From a shortcut menu you can add tic marks, edit the style of the frame bar, and flip the tic marks and labels of the frame bar so they are displayed on the opposite side of the bar.

- The frame bar is automatically updated when you update the concept model.

- Frame bars can be created for all drafting views except Drawing views, Broken views, Revolved Section views, or Unfolded Section views,

Where do I find it?

Frame Bar command

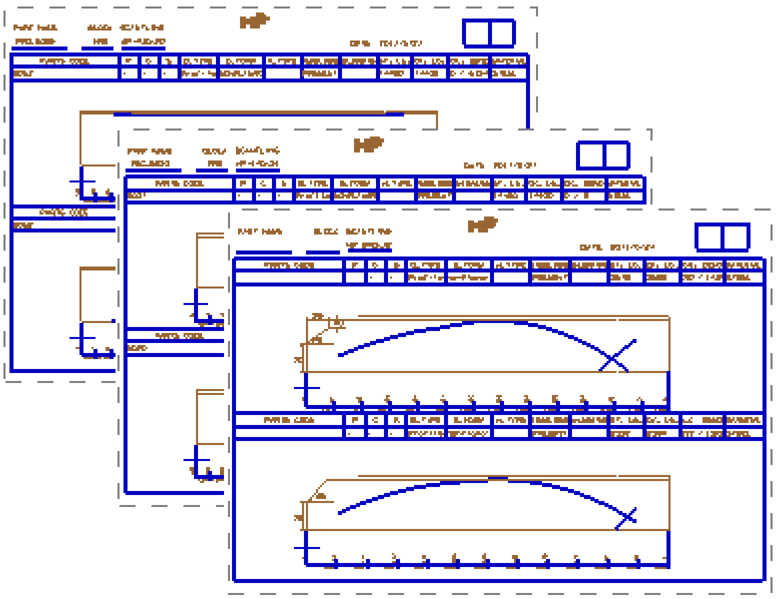
Application	Drafting
Toolbar	**Shipbuilding Drafting® Frame Bar** 
Menu	**Tools® Shipbuilding® Frame Bar**

To set frame bar preferences

Application	Drafting
Toolbar	**Annotation® Annotation Preferences**
Menu	**Preferences® Annotation**
Location in dialog box	**Frame Bar** tab

Inverse Bending Lines Auto-Drawing

Use **Inverse Bending Line Auto-Drawing** to create inverse bending line detail drawings for all profile parts which have inverse bending lines in the current assembly. The drawings are derived from an existing drawing template, and are saved as a booklet of individual files in a specified directory in native NX, and as a single booklet in Teamcenter.



- Each drawing sheet contains up to two separate inverse bending line drawings.
- Profiles that have the same shape and size, but different locations, are all represented by a single drawing.
- The drawing template files, template file notations, and the output location for the completed drawings are specified by options in the Ship Drafting customer defaults.

Where do I find it?

Application	Drafting
Toolbar	**Shipbuilding Drafting® Inverse Bending Lines Auto-Drawing**
Menu	**Tools® Shipbuilding® Inverse Bending Lines Auto-Drawing**

Bending Lines Auto-Drawing customer defaults

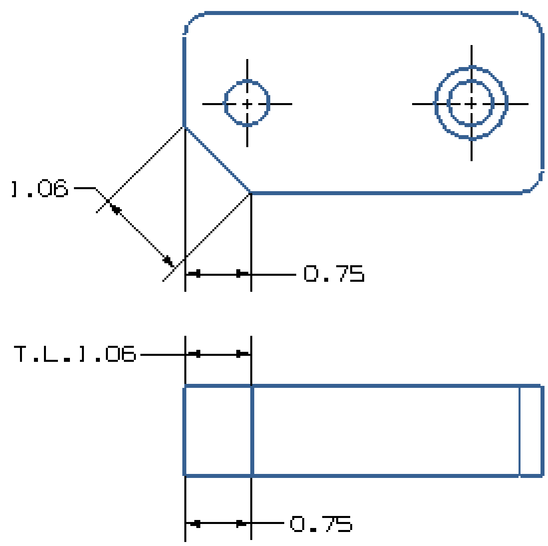
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Ship Drafting® Inverse Bending Lines® Inverse Bending Lines Auto-Drawing** tab

Dimension enhancements

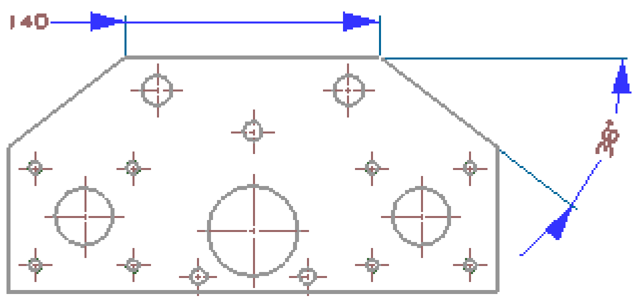
- You can now display the true length of geometry in a projected view. True length is the distance between two picked points. If a single linear edge/curve is selected, the true length is equal to the distance between edge/curve end points.

  Note True length dimensions are available in the Drafting environment for horizontal, vertical, and parallel dimensions only.

  When flagged as true length, the dimension nominal value will be displayed with the flag **T.L.**; this flag text is customizable. You also have the option of displaying the flag text either before or after the dimension.



- The **Arrows in Same Direction** option lets you create a dimension with dimension line arrows that both point in the same direction for linear and angular dimensions.



Note The **Arrows in Same Direction** option is available for Drafting and PMI dimensions.

Where do I find it?

True Length options in the **Annotation Preferences** dialog box

Application	Drafting
Toolbar	**Annotation® Annotation Preferences** 
Menu	**Preferences® Annotation**
Location in dialog box	**Dimensions** tab® **True Length** group® True Length List **Dimensions** tab® **True Length** group® **Text** box

True Length option in the **Dimension Style** dialog box

Application	Drafting
Toolbar	**Dimension®** click any dimension type ® **Dimension Style**  on the **Dimension** dialog bar **Drafting Edit® Edit Style**  ® select one or more annotation objects
Menu	**Edit® Style®** select one or more annotation objects
Location in dialog box	**Dimensions** tab® **True Length** group® True Length List **Dimensions** tab® **True Length** group® **Text** box

Arrows in Same Direction option

Application	Drafting and PMI
Toolbar	In the Drafting application: **Annotation® Annotation Preferences** 
Menu	In the Drafting and PMI applications: **Preferences® Annotation**
Location in dialog box	**Dimensions** tab® Dimension Placement list® **Manual Placement – Arrows in Same Direction** 

Sheet Metal

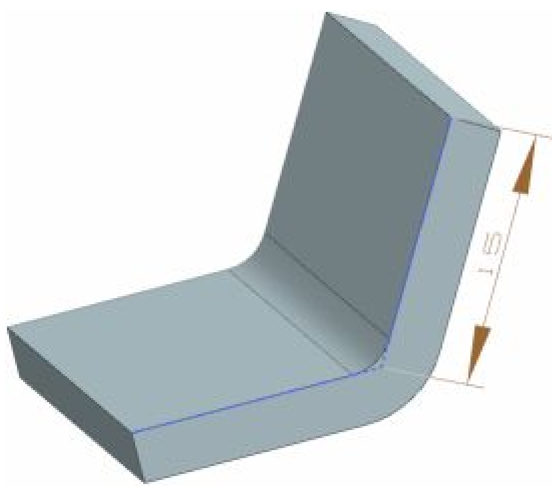
NX Sheet Metal

Flange length measurement enhancements

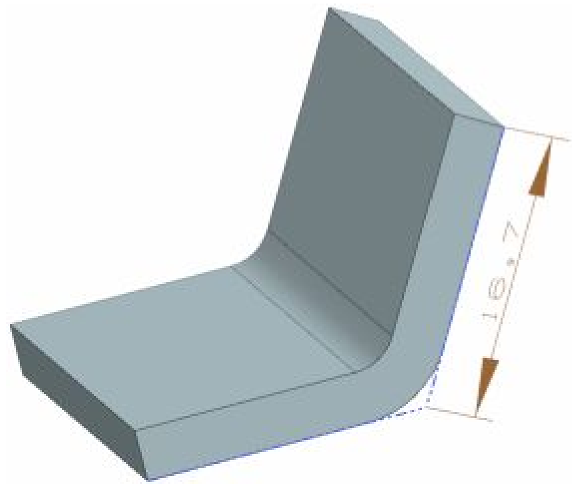
What is it?

The **Inside** and **Outside** flange length options are now specified in a manner that is standard in the straight brake industry. A new **Web** flange length option is added to measure the distance from the end of the adjacent bend to the end of the flange.

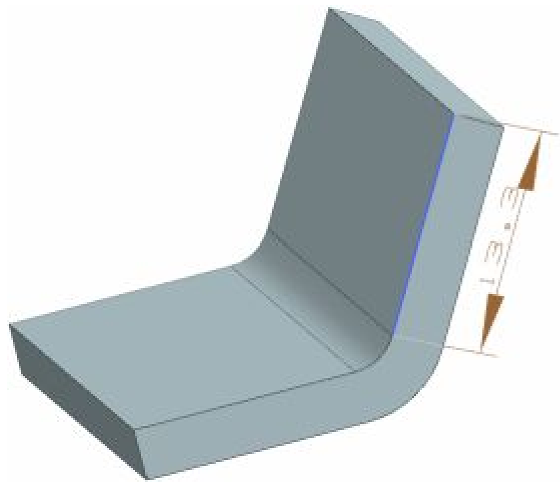
- The **Inside** flange length option now measures the distance from the virtual intersection of the tab faces that are adjacent to the inner bend face, to the top of the flange.



- The **Outside** flange length option now measures the distance from the virtual intersection of the tab faces that are adjacent to the outer bend face, to the top of the flange.



- The **Web** length option measures the distance from the bend tangent line to the top of the flange.



Note This enhancement works for flanges created in NX 8 and later.

Why should I use it?

In previous versions, if the angle of the flange was a value other than 90 degrees, the resulting flange length was not easy to determine and it was not measurable using methods that are commonly employed in the sheet metal industry. You can now accurately measure and dimension flange lengths without using other techniques, such as constructing geometry, to get accurate dimensions.

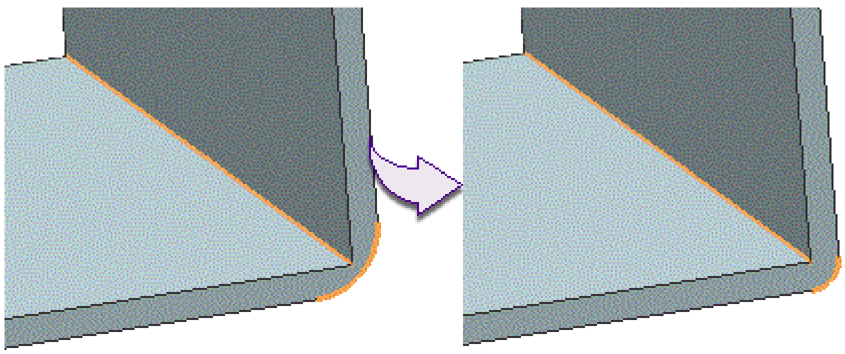
Where do I find it?

Application	**NX Sheet Metal**
Toolbar	**NX Sheet Metal® Flange** 
Menu	**Insert® Bend® Flange**
Location in dialog box	**Flange Properties** group® **Length Reference** list® **Inside** or **Outside** or **Web**

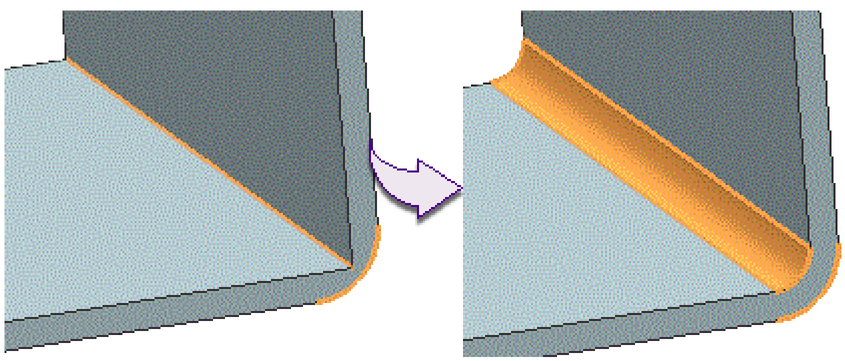
Convert to Sheet Metal enhancement

What is it?

You can now convert imported solid models that have a sharp edge on one side and a bend on the other side, to sheet metal models. To retain the sharp edge, select the **Maintain Zero Bend Radius** check box in the **Convert to Sheet Metal** dialog box. If you do not select this check box, NX converts the sharp edge to a bend, using the bend radius value set for the **Bend Radius** NX Sheet Metal preference.



Resulting edge with Maintain Zero Bend Radius selected



Resulting edge with Maintain Zero Bend Radius not selected

Where do I find it?

Maintain Zero Bend Radius check box

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Convert Drop-down** list® **Convert to Sheet Metal** 
Menu	**Insert**® **Convert**® **Convert to Sheet Metal**
Location in dialog box	**Settings** group® **Maintain Zero Bend Radius** check box

Bend Radius NX Sheet Metal preference

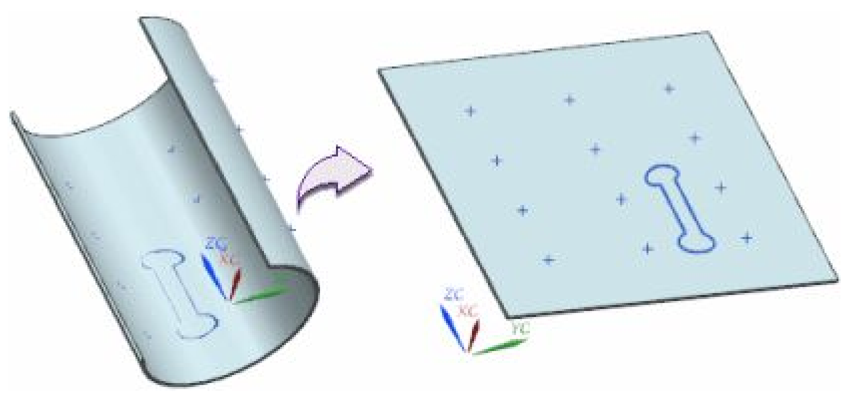
Application	NX Sheet Metal
Menu	**Preferences**® **NX Sheet Metal**
Location in dialog box	**Part Properties** tab® **Global Parameters** group® **Bend Radius**

Unbend enhancements

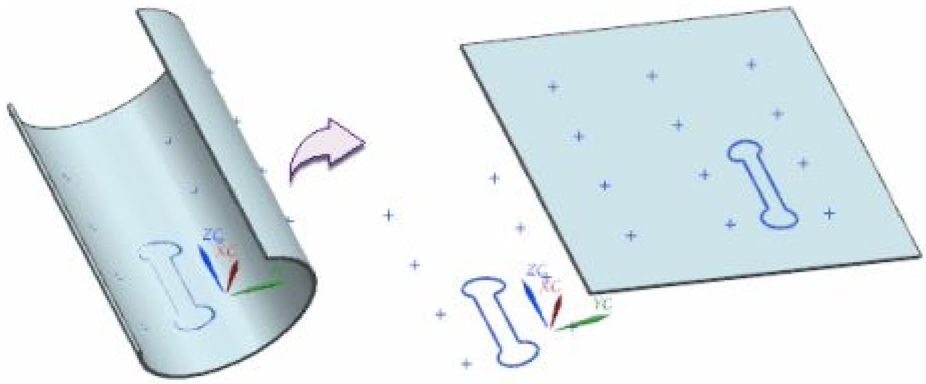
What is it?

You can now transform specified curves and points on a bend face to their flattened state, by using the **Select Curve or Point** option in the **Unbend** dialog box. For example, you can flatten a pattern of points on a lofted flange, as shown.

Use the **Hide Original Curves** check box to control whether to display or hide the original curves and points in the output.



With Hide Original Curves selected



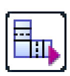
With Hide Original Curves not selected

Why should I use it?

These unbend enhancements let you selectively flatten a set of curves or points in a particular bend region while you are modeling the sheet metal body. You can use these curves and points later to create holes and cutouts. The **Flat Solid** command can produce similar results, but you typically create flat solid bodies and flat patterns after you finish modeling the sheet metal body.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Form Drop-down** list® **Unbend** 
Menu	**Insert**® **Form**® **Unbend**
Location in dialog box	**Settings** group® **Hide Original Curves** check box



Export Flat Pattern

Use the **Export Flat Pattern** command to export flat pattern data to a DXF file or a Trumpf GEO file from within the NX Sheet Metal application. This command replaces the **Export Trumpf GEO File** command, which supported exporting flat pattern data to Trumpf GEO files only. Now you can export flat pattern data to both DXF and GEO files.

Flat pattern data can include curves, annotations, and callouts. Only curves are supported for export. Depending on its type, each flat pattern curve lies on a specific layer in the exported DXF file.

You can assign each layer in the exported DXF file to the required process such as etching, cutting, punching, bending, and so on.

Why should I use it?

The **DXF** option in the **Export Flat Pattern** command provides more sheet metal–specific options than are available when you choose **File→Export→DXF/DWG**. You can more easily select geometry and filter items to export.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**Flat Pattern Drop-down** list® **Export Flat Pattern**
Menu	**Insert**® **Flat Pattern**® **Export Flat Pattern**
Location in dialog box	**Type** group list® **DXF** or **Trumpf GEO**

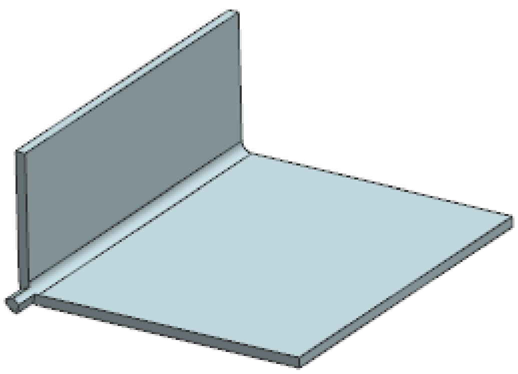
Tab enhancements

What is it?

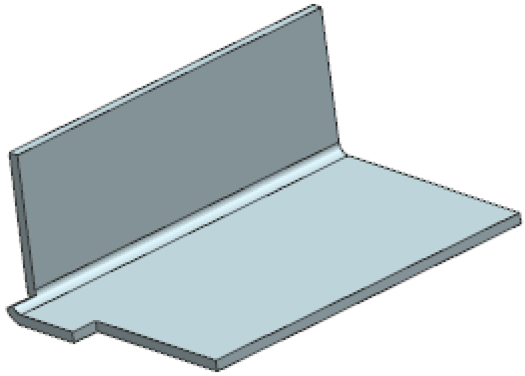
When you use the **Tab** command, you can now create secondary tabs by adding material to bend regions when they are in the unbent state.

You can create secondary tabs on the following areas:

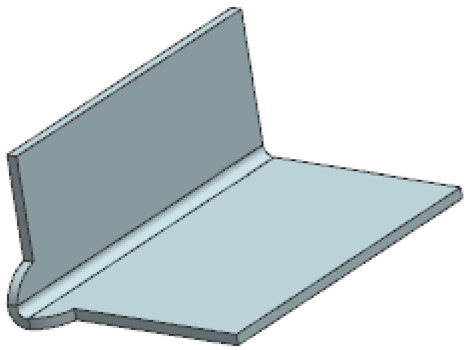
- On bend



- On bend and adjacent face



- On bend and two adjacent faces



Why should I use it?

These enhancements enable you to add material across established bend regions.

Where do I find it?

Application	**NX Sheet Metal**
Prerequisite	A bend region must be present in the work part.
Toolbar	**NX Sheet Metal**® **Tab** 
Menu	**Insert**® **Tab**
Location in dialog box	**Type** group® **Secondary**

Flange face enhancements

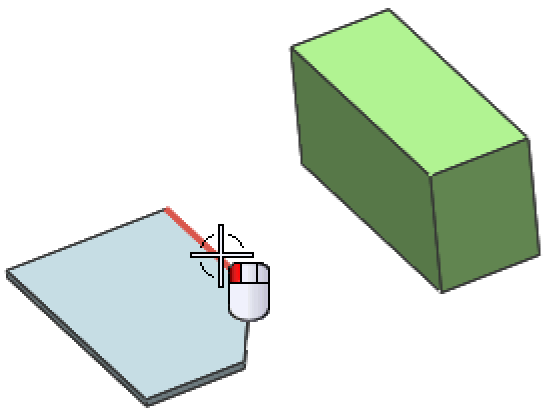
What is it?

You can now select a planar face or datum plane to control the shape and position of a flange using the new **Match Face** and **Specify Plane** options. In previous releases, you could only select an edge to specify the location of a flange.

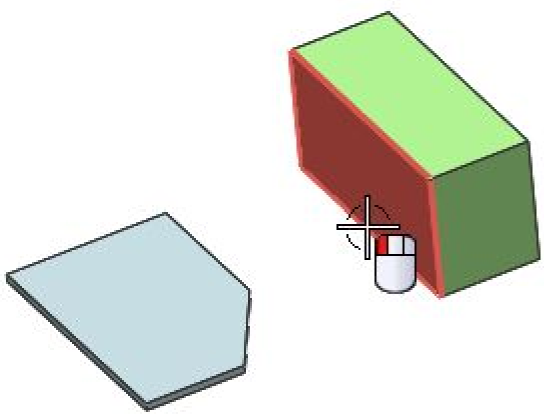
After you select a plane or planar face as input, NX does the following:

- Extends or limits the flange to the selected plane or face.
- Adjusts the bend angle to that of the selected plane or face.

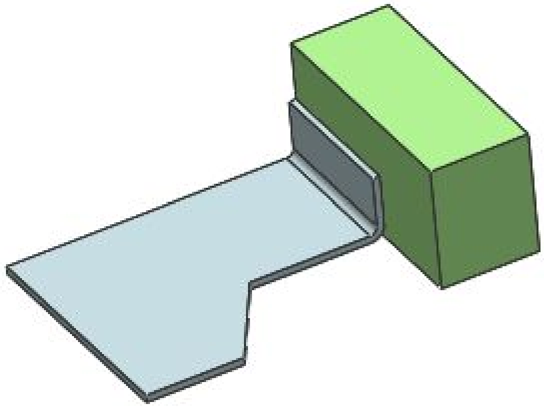
In this example, a flange that is the full width of the tab edge is extended to match the face of another solid body.



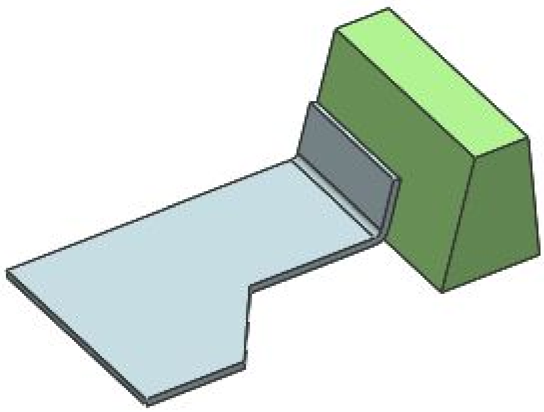
Flange edge selected



Face to match selected



Resulting flange



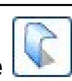
The flange is associative to the matched face, so if the face changes, the flange changes with it.

Note This enhancement works for flanges created in NX 8 and above.

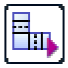
Why should I use it?

The ability to create associative flanges is especially useful in the context of an assembly, where you can use existing geometry to control the extent and angle of a flange. Bracket-type features are a typical use for this enhancement.

Where do I find it?

Application	**NX Sheet Metal**
Prerequisite	The **Specify Plane** option is available only when you select the **Until Selected** option in the **Match Face** list.
Toolbar	**NX Sheet Metal® Flange** 
Menu	**Insert® Bend® Flange**
Location in dialog box	**Flange Properties** group® **Match Face** list

Aerospace Sheet Metal



Export Flat Pattern

Use the **Export Flat Pattern** command to export flat pattern data to a DXF file or a Trumpf GEO file from within the Aerospace Sheet Metal application.

Flat pattern data can include curves, annotations, and callouts. Only curves are supported for export. Depending on its type, each flat pattern curve lies on a specific layer in the exported DXF file.

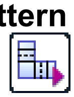
You can assign each layer in the exported DXF file to the required process such as etching, cutting, punching, bending, and so on.

Why should I use it?

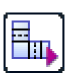
The **DXF** option in the **Export Flat Pattern** command provides more sheet-metal-specific options than are available when you choose **File→Export→DXF/DWG**. You can more easily select geometry and filter items to export.

Because the GEO file format is accepted by Trumpf bending machines and related software, you can transmit data from NX to Trumpf bending machines without the need for general translators such as IGES and STEP. Transmitting data directly into Trumpf machines helps prevent errors and saves time.

Where do I find it?

Application	Aerospace Sheet Metal
Toolbar	**Flat Pattern Feature Drop-down** list® **Export Flat Pattern** 
Menu	**Insert**® **Flat Pattern**® **Export Flat Pattern**
Location in dialog box	**Type** group list® **DXF** or **Trumpf GEO**

Flexible Printed Circuit Design



Export Flat Pattern

Use the **Export Flat Pattern** command to export flat pattern data to a DXF file or a Trumpf GEO file from within the Flexible Printed Circuit Design application.

Flat pattern data can include curves, annotations, and callouts. Only curves are supported for export. Depending on its type, each flat pattern curve lies on a specific layer in the exported DXF file.

You can assign each layer in the exported DXF file to the required process such as etching, cutting, punching, bending, and so on.

Why should I use it?

The **DXF** option in the **Export Flat Pattern** command provides more FPCD-specific options than are available when you choose **File→Export→DXF/DWG**. You can more easily select geometry and filter items to export.

Because the GEO file format is accepted by Trumpf bending machines and related software, you can transmit data from NX to Trumpf bending machines without the need for general translators such as IGES and STEP. Transmitting data directly into Trumpf machines helps prevent errors and saves time.

Where do I find it?

Application	Flexible Printed Circuit Design
Toolbar	**Flat Pattern Drop-down** list® **Export Flat Pattern**
Menu	**Insert**® **Flat Pattern**® **Export Flat Pattern**
Location in dialog box	**Type** group list® **DXF** or **Trumpf GEO**

Printed Circuit Design

PCB Exchange

Baseline EDMD baseline schema support

What is it?

You can now import and export *IDX* files in the **PCB Exchange** application. The *IDX* are neutral formats based on STEP standards and are compatible with different ECAD programs. The STEP standards are standards for the exchange of product model data. *IDX* stands for interdomain design exchange, and *EDMD* for electrical design mechanical design

The *IDX* files contain physical design information for electronic circuit boards.

Incremental import and export operations are not supported.

Where do I find it?

Application	**PCB Exchange**
Toolbar	**PCB Exchange** ® **Import ECAD Model** or **Export ECAD Model**
Menu	**PCB Exchange** ® **Import ECAD Model** or **Export ECAD Model**

New board conductivity calculation algorithm

What is it?

You can now use a new board thermal conductivity calculation algorithm. The *equivalent* algorithm provides a calculation point averaged conductivity based on a pixel discretization of the board conductive and non conductive layers.

Previously, NX used the *discretized* algorithm that was named **Automatic**. This algorithm is now named **Discretized**.

Why should I use it?

Compared to the discretized algorithm, the equivalent algorithm provides a faster alternative to calculate the orthotropic board conductivity.

Where do I find it?

Application	**PCB Exchange**
Toolbar	**PCB Exchange** ® **Board Mesh And Thermal Settings**
Menu	**PCB Exchange** ® **Board Mesh And Thermal Settings**
Location in the dialog box	**Thermal Conductivity** group ® **Algorithm**® **Discretized**

Usability enhancements in PCB Exchange dialog boxes

What is it?

The following dialog boxes have been reorganized for easier access:

- **Board Mesh and Thermal Settings**
- **Default Component Mesh and Thermal Settings**

The order of the options in the dialog boxes changed to improve the workflow.

In the **Create ESC Solution** dialog box, you can now set the node and element IDs for the first node and element in the mesh.

Where do I find it?

Application	**PCB Exchange**
Toolbar	**PCB Exchange** ®  **Board Mesh and Thermal Settings**,  **Default Component Mesh and Thermal Settings**, and  **Create ESC Solution**
Menu	**PCB Exchange** ® **Board Mesh and Thermal Settings**, **Default Component Mesh and Thermal Settings**, and **Create ESC Solution**

New entities created from the Create ESC Solution command

What is it?

When you use the **Create ESC Solution** command, PCB Exchange creates new entities in Advanced Simulation to improve the workflow.

Now, the following entities are created:

In the FEM file

- Meshes representing the board and components.
- A 2D Shell mesh collector of type **PCB Stack**.
- A **PCB Stack** physical property which defines stacks of layers and vias in the printed circuit board.
- **PCB Layer** and **PCB Vias** modeling objects which define materials and distributions in the printed circuit board.
- Fields that model the orthotropic board thermal conductivity.

In the Simulation file

- A **Printed Circuit Board** simulation object which defines friction properties for the printed circuit board.
- **PCB Component** simulation objects which represent components, connect them to the board, define dissipation, and produce reports.

Previously, for each component, NX PCB Exchange created one **Thermal Coupling** simulation object that connected the component to the printed circuit board and one **Thermal Load** load. It also created a **Property Override** simulation objects that modeled orthotropic board conductivity.

Why should I use it?

PCB Exchange now creates less boundary conditions when creating a simulation file. You can now easily access and modify the orthotropic board conductivity entities.

Where do I find it?

Application	**PCB Exchange**
Toolbar	**PCB Exchange ® Create ESC Solution**
Menu	**PCB Exchange ® Create ESC Solution**

PC Design Navigator

What is it?

Use the new **PC Design Navigator** command to identify, access, and edit ECAD parameters of the following *Printed Circuit Assembly* (PCA) entities:

1. Boards
2. Components
3. Areas
4. Holes

For the previous PCA entities you can:

- Select one or more components and observe the highlighted component in the graphics window.
- Order the list of entities by any of their ECAD attributes.
- Modify the ECAD attributes of one or more components at once.

Previously, you needed to modify the ECAD properties of each entity individually.

Why should I use it?

The **PC Design Navigator** provides a faster access to entity attributes in a single interface window.

Where do I find it?

Application	**PCB Exchange**
Toolbar	**PCB Exchange ® PC Design Navigator**
Menu	**PCB Exchange ® PC Design Navigator**

Flexible Printed Circuit Design

Export Flat Pattern

Use the **Export Flat Pattern** command to export flat pattern data to a DXF file or a Trumpf GEO file.

Flat pattern data can include curves, annotations, and callouts. Due to technological limitations, only curves are supported for export.

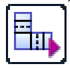
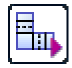
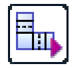
Depending on its type, each flat pattern curve lies on a specific layer in the exported DXF file.

Type of flat pattern curve	Layer number
Bend up center lines	1
Bend down center lines	2
Bend tangent lines	3
Outer mold lines	4
Inner mold lines	5
Exterior curves	6
Interior cutout curves	7
Interior feature curves	8
Center of lightening hole	9
Joggle line between stationary & transition areas	10
Added top curves and points	11
Added bottom curves and points	12
Tool marker	13

You can:

- Select the curves to be exported more easily.
- Assign each layer in the exported DXF file to the required processes such as etching, cutting, punching, bending, and so on.

Where do I find it?

Application	NX Sheet Metal, Aerospace Sheet Metal, Flexible Printed Circuit Design
Toolbar	(NX Sheet Metal) **Flat Pattern Drop-down** list® **Export Flat Pattern**  (Aerospace Sheet Metal) **Flat Pattern Feature Drop-down** list® **Export Flat Pattern**  (Flexible Printed Circuit Design) **Flat Pattern Drop-down** list® **Export Flat Pattern** 
Menu	**Insert**® **Flat Pattern**® **Export Flat Pattern**
Location in dialog box	**Type** group list® **DXF** or **Trumpf GEO**

Mechatronics Concept Designer

Physics Objects Converter

Use the **Physics Objects Converter** command to convert physics objects to motion objects. The conversion makes it possible for the NX Motion Simulation application to interpret a Mechatronics Concept Designer model. While the conversion is automatic, you can choose the objects to be included in the conversion.

Not all physics objects can be converted to motion objects. The following table shows the physics objects that can be converted and their corresponding motion object.

Physics object	Motion object
Rigid Body	Link
Hinge Joint	Revolute Joint
Sliding Joint	Slider Joint
Cylindrical Joint	Cylindrical Joint
Ball Joint	Spherical Joint
Fixed Joint	Fixed Joint
Gear	Two-Joint Coupler
Linear Spring Joint	Spring & Damper
Angular Spring Joint	Bushing
Speed Control	Joint Driver

Each motion object that is created by this conversion has the same name as the physics object on which it is based, unless the physics object has a name that is invalid for motion objects. An invalid name is a name that contains any special characters other than Dash (-) or Underscore (_).

Note This command can only convert the physic objects when you create a new simulation. It can not convert objects and add them to an existing simulation.

Note If your model contains assembly constraints, a warning is displayed during the conversion that gives you the option to convert the constraints in addition to the Mechatronics Concept Designer objects. It is recommended to convert only the Mechatronics Concept Designer objects into motion objects. Converting the constraints can cause an over-constrained model.

Where do I find it?

Application	Motion Simulation
Prerequisite	You must create a new simulation for a work part that contains Mechatronics Concept Designer physics objects.

Replace Assistance

What is it?

Use the **Replace Assistance** dialog box to migrate the physics objects defined in a concept model to a detailed model. You can specify geometry references for each physics object when you replace concept model components with detailed model components. All related physics objects are automatically regenerated in the detailed components after you specify the references.

Why should I use it?

It provides the capability to reassign corresponding references from concept model to detailed model in assembly context.

Where do I find it?

Application	Mechatronics Concept Designer
Prerequisite	You must click **OK** or **Apply** in the **Replace Component** dialog box.
Toolbar	**Assemblies® Replace Component** 
Menu	**Assemblies® Components® Replace Component**

SNAP — New programming tool

What is it?

A new programming tool is added to the NX system for creating custom programs. Simple NX Application Programming (SNAP) is an easy-to-learn programming tool intended for mechanical designers and other typical NX users – not just for programmers. The code is simple, short, readable, and feels “natural” so it is quicker to learn and easier to apply.

SNAP is useful for automating simple processes that you do repeatedly, or for creating simple parts that you use frequently.

SNAP is similar to the GRIP language in its purpose, flexibility, and ease of use. So if you have used GRIP before, you will probably also find SNAP to be a productive tool.

SNAP is based on the Visual Basic (VB.Net) language and is best used with Visual Studio, however you can create some simple programs using the NX Journal Editor.

An example of SNAP code is shown below:

```
myFunction
   length = 8
   width = 4
   half = width/2
   left = Line(-half, 0, —half, length) ‘Left side; Line(x1, y1, x2, y2)
   right = Line(half, 0, half, length)  ‘Right side
   top = Arc(0, length, half, 0, 180)   ‘Top semi-circle; Arc(x, y, r, 01, 02)
   bottom = Arc(0, 0, half, 180, 360)   ‘Bottom semi-circle
   Dim outline As NX.Curve() = {left, right, top, bottom}
   boss = Extrude(outline, 3)
End
```

SNAP and NX Open can easily be used together and you may find that many of your programs will use a mixture of SNAP and NX Open functions. In fact, SNAP can be regarded as a stepping stone that can lead you into the more powerful NX Open programming functionality.

In this first release, SNAP is focused on the creation of simple geometry. Other areas of NX will be covered in future versions.

SNAP is only supported on Windows machines.

Why should I use it?

You can use SNAP to easily automate repetitive NX tasks even if you do not have any previous programming experience.

Where do I find it?

A Getting Started with SNAP guide is provided to introduce you to SNAP and help you learn how to use it as quickly as possible. It also includes a brief introduction to the Visual Basic language.

An API reference manual (.chm file) provides detailed documentation for all of the functions in SNAP, plus many examples.

The guide and reference manual (.chm file) are located in the Programming Tools section of the documentation.

Chapter

5 CAM (Manufacturing)

Manufacturing General

Tool library enhancements

What is it?

The parameters in the NX ASCII Tool library now match the parameters in the NX **Create Tool** dialog box.

New parameters that are added to the ASCII tool library are:

- Milling tool shank
- Coolant through on mills/drills
- Tip angle on mills
- Flute length on user defined mills
- Corner radius on holemaking tools
- Bell angle and diameter on center drills
- Neck diameter and taper angle on taps and reamers
- Boring bar parameters
- Probe tracking points
- Turning parametric holder
- Boring Diameter, Maximum Reach, and Maximum Depth on Turning tools
- X Mount, Y Mount, and Z Mount
- Milling chamfer

New classes that are added to ASCII tool library are:

- Chamfer Mills
- Spherical Mills
- Spot facing tools

New Non Cutting Moves engage parameters are available for all milling tools.

- Ramp Angle
- Diameter (helical)
- Minimum Ramp Length

A new cutting parameter (that is currently used by plunge milling), is available for all milling tools.

- Max Cut Width

The tool classes and parameters are recognized when you export tools to the Tool library or retrieve them.

You can now:

- Search for tools using a string in their description.
- Export tools with no holding system defined.
- Assign a custom suffix number for the libref when exporting or importing a tool or holder. A new **Starting Number for User Librefs** customer default is available.

Note If you customized the out of the box ASCII libraries *tool_database.dat*, you must update your entries to the new format.

When a tool is retrieved, User Defined Events on the tool master in the *library_dialog.prt* are added to the tool.

Why should I use it?

These enhancements help you to maintain complete tool definitions and consistency.

The new customer default will prevent existing libref numbers from being overwritten when a new libref number is created.

Where do I find it?

ASCII Tool library

Application	Manufacturing
Menu	**Insert**® **Tool**
Location in dialog box	**Library** group® **Retrieve Tool from Library** ® **Library Class Selection**® **Search Criteria** dialog box® **Search Parameters** group® **Description**

Starting Number for User Librefs customer default

Application	Manufacturing
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Manufacturing**® **Tools**® **Tool Library** group

Shop Documentation enhancements

What is it?

The following templates are now available:

- HTML templates. These are templates that you can customize in Excel.
- TEXT templates. These are templates that you can customize in TCL.

In order to create Shop Documentation, you must first select an object in the **Operation Navigator**. Both types of templates process objects that are selected in the **Operation Navigator**.

Note Pre-NX 8 templates are not visible in NX because you can use the default templates to create a report of machining operations from any of the four **Operation Navigator** views. Pre-NX 8 templates generate reports only for a single view. If you want to use pre NX 8 templates, remove the comment sign (#) in the *shop_doc.dat* file . The file is located in *MACH\resource\shop_doc*.

When you select an Excel-html template in the **Shop Documentation** dialog box, you will create a document that can be displayed in Microsoft Excel™ or web browsers.

The Excel files used to create the Excel-html templates are stored in the *MACH/resource/shop_doc/excel_templates* folder; they can be further customized to produce new templates. Each Excel file has two sheets. The second sheet contains MOM variables and descriptions that you can copy and paste into the appropriate cells in the first sheet. This enables you to edit the templates even if you are not familiar with MOM.

Why should I use it?

You can customize the template and its contents to better control the **Shop Documentation** output based on the specific requirements of the shop.

Where do I find it?

Application	Manufacturing
Toolbar	**Operations** toolbar
Location in dialog box	**Shop Documentation**  dialog box® **Report Format** list

CAM Express startup

What is it?

You can now launch CAM Express from the **Start** menu on your computer as a separate product from NX.

Why should I use it?

Launch CAM Express as a separate product to explicitly differentiate it from NX.

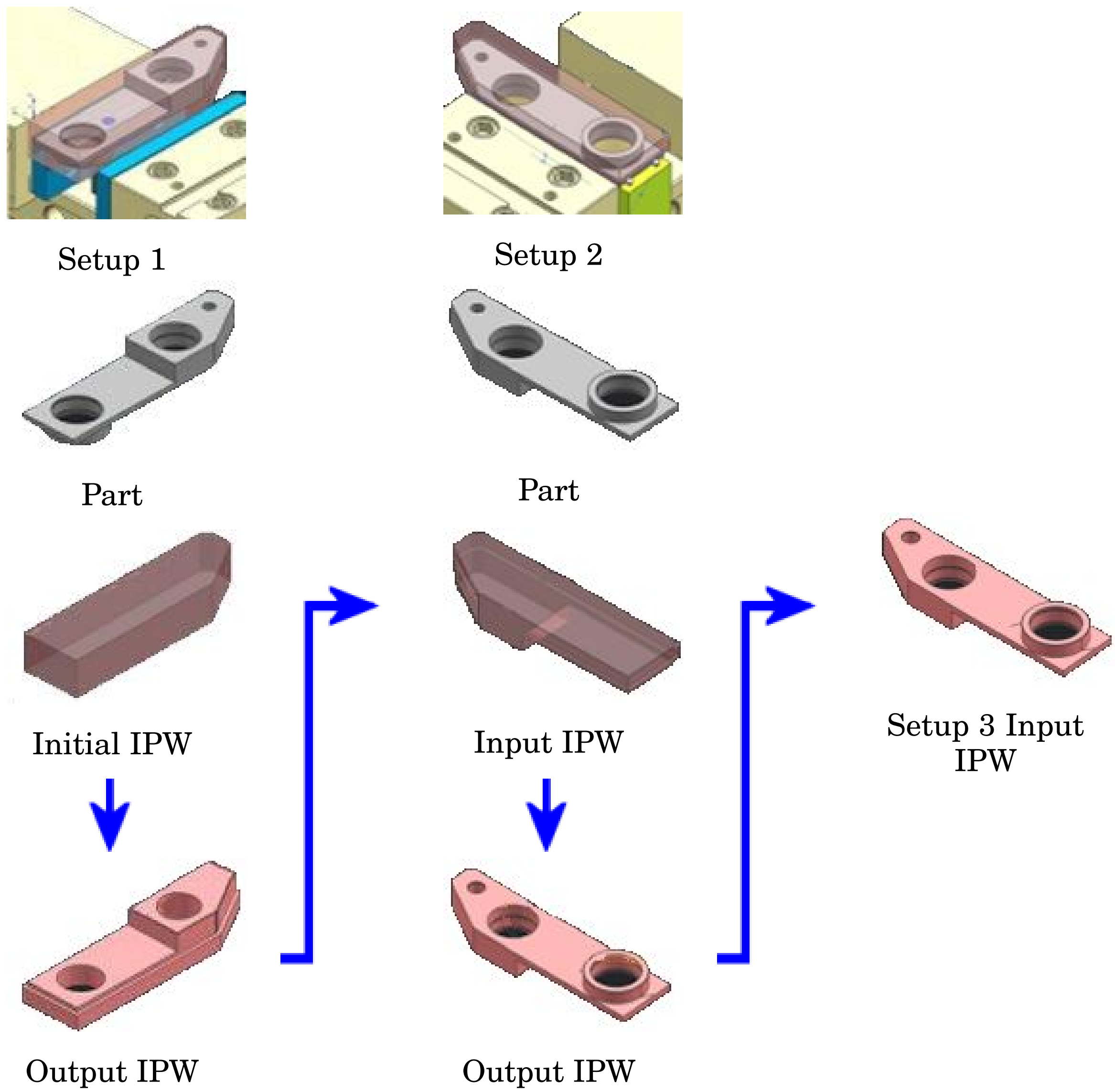
Where do I find it?

Application	Manufacturing
Menu	**Start® All Programs® Siemens NX 8.0® Manufacturing® CAM Express 8**

Transferring an IPW across multiple setups

What is it?

You can now transfer an In Process Workpiece (IPW) from one manufacturing setup to the next manufacturing setup. NX automatically positions the IPW within and across setups.



You can establish the relationship between two setups for the IPW transfer. The IPW is transferred from the previous part position to the new part position and the two are associative. You can view the relationship in the **Blank Geometry** dialog box.

Why should I use it?

Transfer an IPW from one setup to the next setup when a part cannot be machined in a single orientation and you need to either reposition the part or refixture it.

Where do I find it?

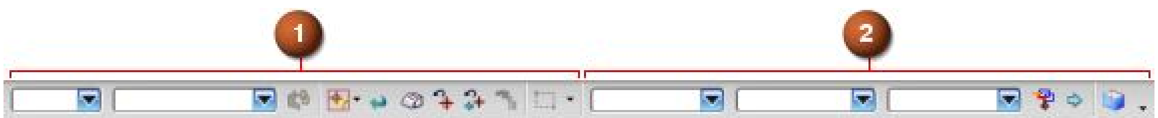
Application	Manufacturing
Prerequisite	The selected part geometry must reside in the same assembly component.
Location in dialog box	**WORKPIECE** group® **Blank Geometry**® **IPW**® **Select Source for IPW**

Geometry selection in Manufacturing

What is it?

Face and body selection in Manufacturing is now consistent with face and body selection in Modeling. This is true for all interactions where you select geometry *except* for boundary selection. You can now do the following:

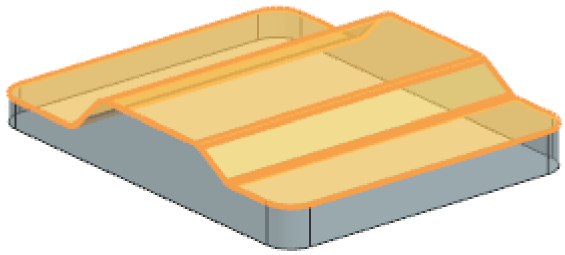
- Use the Selection filters 1 on the Selection bar to filter face and body selection, and use the Selection Intent rules 2 to select multiple objects.



- Append a new set of objects with a different selection rule to the current selection. You must click **Add New Set** in a selection dialog box for CAM faces and bodies.

- Assign custom data for each set of faces collected by a selection rule.

- Verify your selections by the same object highlighting that you see in modeling.

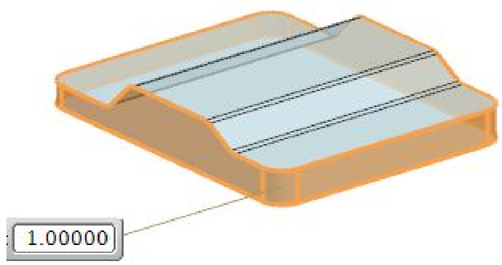


Caution Try to anticipate possible design changes when you specify selection intent. If the geometry changes, the selection intent rules may modfiy your operation by adding or removing faces. For example, if you specify **Tangent Faces** to collect all faces of a blended pocket, and the designer later adds a blend at the pocket opening, adjacent faces are added to the collection. Typically, in such situations, there is an alternative rule that is better-suited to collect faces according to your intent, such as **Feature Faces**, **Region Faces**, and so on.

You can also select individual faces, just as you did in earlier releases.

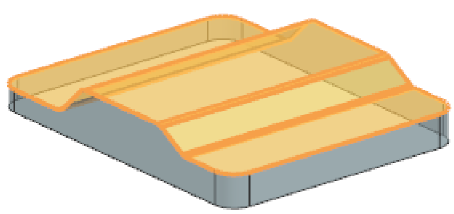
The following example shows different custom stock assigned to sets of work piece faces.

Set	Number of Items	Custom Data
1	8	Yes
2	9	No
3	1	Yes



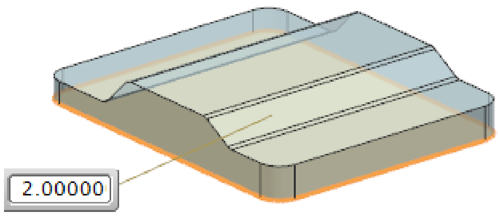
Set 1 is assigned custom data. The on-screen input box shows a **Part Offset** value of 1.0 mm.

Face Rule is set to **Tangent Faces**; the entire perimeter of the part is selected with a single click.



Set 2 has the default stock.

Face Rule is set to **Tangent Faces**; the entire set of upper faces of the part is selected with a single click.



Set 3 has custom data.

The on-screen input box shows a **Part Offset** value of 2.0 mm.

For the lower face, **Face Rule** is set to **Single Face**. With this rule, you can select as many faces as you require, one by one.

Legacy part updates

- When you open legacy parts, all objects with identical custom data that have the same stock value are grouped into one set.

- Custom data for gouging is no longer supported.

Why should I use it?

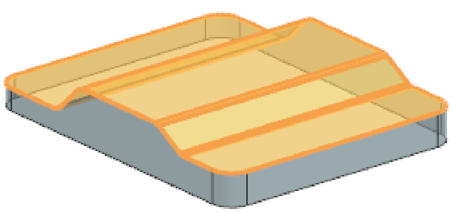
Use selection rules to reduce the number of clicks that you need to select faces and bodies.

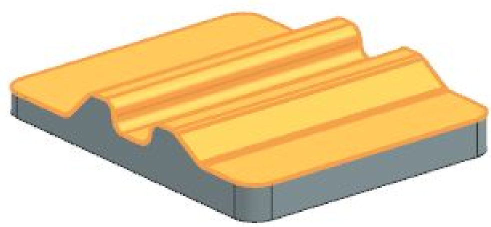
When you modify the selected geometry, the selection rule is reevaluated and faces or bodies that meet the rule are collected in the set automatically.

In the following example, the upper faces of the body are modified. Because the **Face Rule** value for set 2 is **Tangent Faces**, the additional faces are automatically selected. The new faces are assigned the **Part Offset** value for the set.

Set	Number of Items	Custom Data
1	8	Yes
2	9	No
3	1	Yes

Set	Number of Items	Custom Data
1	8	Yes
2	17	No
3	1	Yes





Dynamic machine tool positioning

What is it?

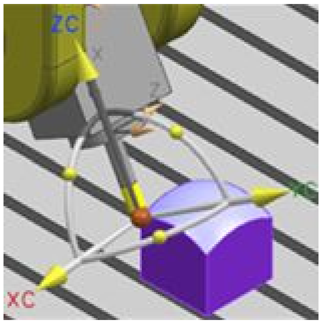
Your machine tool model is dynamically positioned in your Manufacturing setup when you use the CSYS tool to:

- Define a tool axis in a fixed-axis or variable-axis operation.
- Edit a milling, drilling, or probing tool.

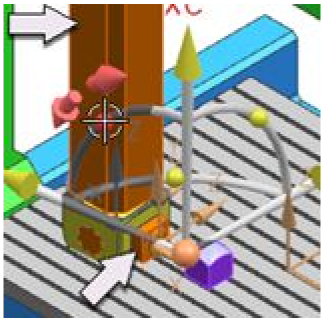
Note There are two customer defaults to control the dynamic display of the machine tool. Out of the box the dynamic machine tool is on for setting the tool axis and off for editing tools.

If you move a machine axis to an over-travel position, you see the following graphical feedback:

- The machine tool component that has over-travelled is highlighted.
- The drag indicators for the handle that caused the over-travel are highlighted.



Machine tool components are moved to match the tool axis



Graphic alerts for over-travel

Why should I use it?

You should use dynamic simulation of machine tool components whenever you need to visually check machine tool limits.

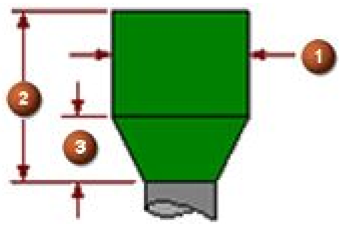
Where do I find it?

Application	Manufacturing
Prerequisite	A library machine must be present in the setup and it must be properly configured.

Shank definition for milling and drilling tools

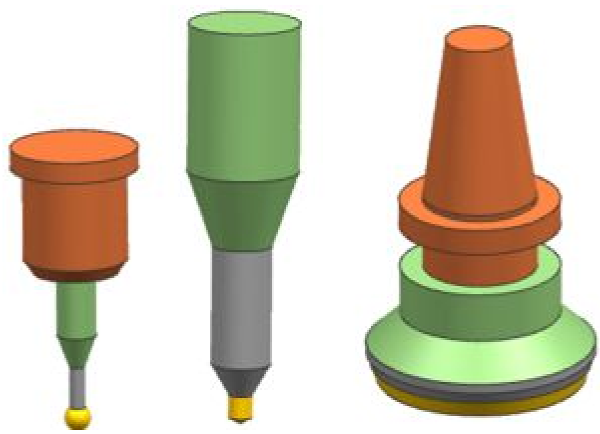
What is it?

When you create milling and drilling tools, you can define the shank between the cutting tool and the tool holder. You can specify the following shank parameters:



1. **Shank Diameter**
2. **Shank Length**
3. **Shank Taper Length**

There is a tapered transition from the tool to the shank diameter. The shank is included in the tool definition in the tool library.



The previous shank diameter found on tools such as T Cutters is now called **Neck Diameter**.

Why should I use it?

You can define a more complete and accurate representation of your cutting tool.

Where do I find it?

Application	Manufacturing
Menu	**Insert**® **Tool**
Toolbar	**Insert**® **Create Tool**
Location in dialog box	**Tool Subtype** group® **Mill Tool Parameters** dialog box® **Shank** tab

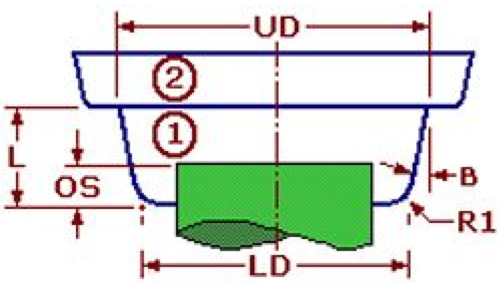
Taper definition for tool holder steps

What is it?

An **Upper Diameter** parameter is added to the tool holder parameters. NX can calculate the **Upper Diameter** value from the **Taper Angle** and **Length** values or you can specify the **Upper Diameter** value directly.

The **Diameter** parameter in previous releases is now called **Lower Diameter**.

You can specify the following parameters when defining a tool holder:



(L) **Length**

(OS) Tool Insertion **Offset**

(UD) **Upper Diameter**

(LD) **Lower Diameter**

(B) **Taper Angle**

(R1) **Corner Radius**

(1) Step 1

(2) Step 2

Why should I use it?

The improved tool holder interface makes it easier to define tapered holder steps that align with, and are stacked with, adjacent steps.

Where do I find it?

Application	Manufacturing
Menu	**Insert**® **Tool**
Location in dialog box	**Create Tool** dialog box® **Tool Subtype**® Milling Tool dialog box **Holder** tab

Operation parameters inherited from the tool

What is it?

You can now specify the following Operation Parameters when creating most milling tools.

- **Ramp Angle**
- **Helical Diameter**
- **Min Ramp Length**
- **Max Cut Width**

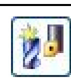
In previous versions, you could only specify these parameters within the operation.

Note Out of the box, inheritance of these parameters is turned off in the operations.

Why should I use it?

Because the parameters are saved with the tool, they can be inherited into the Cutting Parameters and Non-Cutting Moves of operations that use the tool. This is most useful for example, for tools such as inserted mills where the engage ramp angle and diameter are a characteristic of the tool.

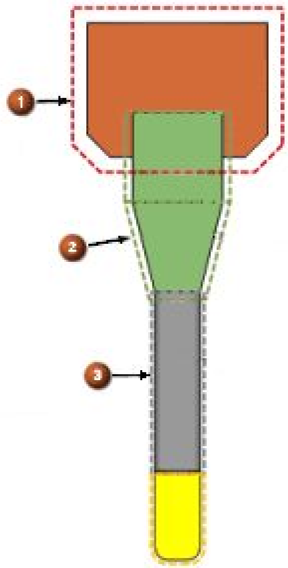
Where do I find it?

Application	Manufacturing
Menu	**Insert**® **Tool**
Toolbar	**Insert**® **Create Tool** 
Location in dialog box	**Tool Subtype** group® **Mill Tool Parameters** dialog box® **More** tab® **Operation Parameters** group

Tool clearance enhancements

What is it?

Use the new tool and holder clearance parameters to gain additional clearance control over different segments of the tool.



1. **Tool Holder**
2. **Tool Shank**
3. **Tool Neck**

Clearance parameters are used in several places, including operation collision checking, **Operation Navigator** gouge checking, and tool axis tilting.

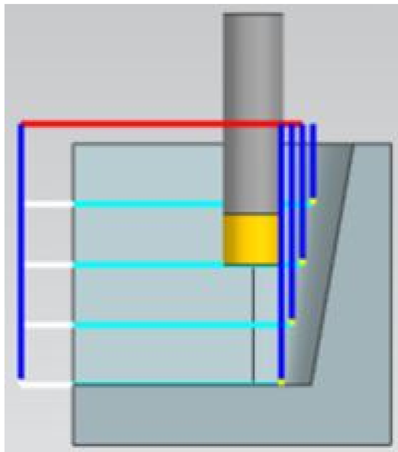
Where do I find it?

Application	Manufacturing
Menu	**Insert**® **Create Operation**
Location in dialog box	**Path Settings** group® **Cutting Parameters**® **More** tab® **Clearance** group

Flute length checking

What is it?

The new **Check Flute Length and Depth per Cut** customer default lets you check the nominal depth of cut against the flute length of the tool in many milling operations.



If this customer default is selected, and the flute length of the tool is less than the nominal programmed cut depth, a warning is issued with an option to continue.

Why should I use it?

You can validate that the tool flute length is long enough to cut to the specified depth per cut.

Where do I find it?

Application	Manufacturing
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Manufacturing**® **Tools**® **Validation** tab® **Milling Operations** group® **Check Flute Length and Depth per Cut**

Manufacturing Milling

Blank Geometry enhancements

What is it?

The **Blank Geometry** dialog box has been updated and the following new options are available for all milling operations that support blank geometry.

Bounding Block

Similar to the legacy **Auto Block** option. You now have an orientation list, from which you can select **MCS** or **Specify CSYS**.

Bounding Cylinder

Encloses the part outline in a circle and extrudes the shape along the current MCS to completely enclose the part.

Note **Bounding Cylinder** provides an optimal blank for cylindrical parts. You may need to increase the blank size for rectangular parts.

Part Outline

Extrudes the part outline to completely enclose the part.

Part Convex Hull

Connects the extreme points of the part outline to create a convex hull and extrudes the shape to completely enclose the part.

IPW – In Process Workpiece

Lets you specify the source for the In-process workpiece.

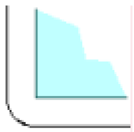
For the **Bounding Cylinder**, **Part Outline**, and **Part Convex Hull** options, you can:

- Accept the default extrusion direction, which is parallel to the ZM-axis of the MCS, or specify a different direction vector.

- Increase the top or bottom limits.

- Offset the circle radius, part outline, or convex hull outline.

  The part and convex hull offset outlines roll around corners.



Where do I find it?

Application	Manufacturing
Toolbar	**Insert**® **Create Geometry**
Menu	**Insert**® **Create Geometry**
Location in dialog box	**Geometry Subtype** group® **Workpiece** or **MILL_GEOM** subtype® **Workpiece** dialog box® **Geometry** group® **Specify Blank** ® **Blank Geometry** dialog box® **Type** group

Feed rate setting improvements

What is it?

Feed rate settings in the **Feeds and Speeds** dialog box for milling operations are improved in the following ways.

Rapid motion output mode

In the **Feed Rates** group, under **Rapid**, you can set **Output** to **G0 – Rapid Mode** or **G1 – Feed Mode**.

Motion types

Under **More**, you can specify that motions use the rapid output mode or a percent of the cut feed rate. For example, from the **Approach** list, you can choose from the following:

- **none**
- **ipm**
- **ipr**
- **Rapid**
- **%Cut**

Zero feed rates

Zero feed rates are no longer permitted with the new rapid and percent of cut feed rate options.

Where a zero feed rate previously implied a rapid move, you can specify **Rapid** to use the current rapid output mode.

Where a zero feed rate previously implied the cut feed rate, you can specify a percent of the cut feed rate.

Operation Navigator

When you edit feed rates in the **Operation Navigator**, if multiple operations of the same type are selected, all applicable feed rates are available for editing.

Calculation of related values

In the **Feeds and Speeds** dialog box, each of the **Surface Speed**, **Feed per Tooth**, **Spindle Speed**, and **Cut Feed** boxes has a corresponding calculate button. Each button recalculates the other three values. If only one button is active, the values are calculated when you click **OK**. If none of the buttons are active, all of the values are up-to-date.

Set Machining Data

When you click **Set Machining Data**:

- Feed rates are set as a percent of the cut feed. Feed values in the feeds and speeds database are preserved.
- For user-defined milling tools with tracking points, the diameter at the selected tracking point is used to calculate the spindle speed or feed rate.

Part update

Parts from previous versions of NX are automatically updated. Zero feed rates are converted either to rapid motion or 100% of cut feed, depending on the rule for the zero value in the earlier release.

Note The preceding changes do not apply to **Wire EDM** feed rates.

Why should I use it?

In non-interpolated rapid mode, some machines may not move in a direct path; they might make a dogleg motion. If you need more control, for example to avoid potential collisions, you can now specify interpolated motion.

Where do I find it?

Application	Manufacturing
Prerequisite	Open any dialog box that contains **Feeds and Speeds** , or, in the **Operation Navigator**, select one operation or multiple operations of the same type.

Variable-axis profiling enhancements

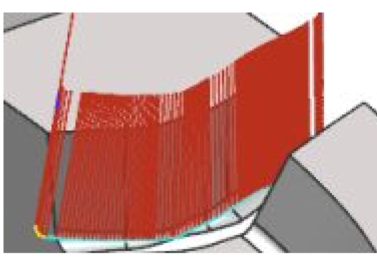
What is it?

You now have the following options to control cutter motion and orientation in variable-axis profiling.

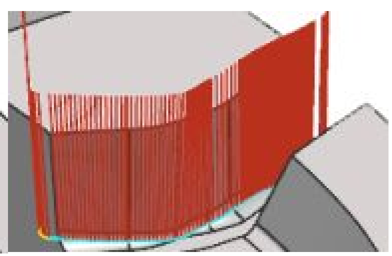
Automatic Uses the same positioning method as in pre-NX 8 releases. Legacy tool paths are updated to this option.

Align to Edges Aligns the tool tangent to straight edges between the top and bottom of the wall.

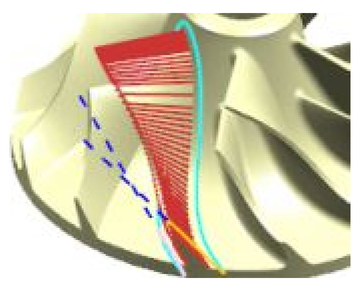
Swarf Base UV Aligns the tool to the UV rulings of the wall.



Automatic



Align to Edges



Swarf Base UV

Why should I use it?

Use a variable-axis profile operation to swarf-cut walls. To get the best result, you must adjust the tool alignment in the lead and lag direction based on the wall geometry.

- Use the **Swarf Base UV** option when you have a ruled surface with proper UV alignment.
- Use the **Align to Edges** option when there are straight edges that guide the alignment properly.
- Use the **Automatic** option in all other cases.

Where do I find it?

Application	Manufacturing
Menu	**Create Operation® mill_multi-axis® VARIABLE_CONTOUR® Drive Method® Contour Profile**
Location in dialog box	**Axis** group® **Axis** list

Multi Blade Milling

Multi Blade — Interpolate Vector

Use the **Interpolate Vector** tool axis option to control tool alignment behavior. You can edit orientation at system points and add or manipulate user points along the lowest and highest cut levels. NX interpolates along and between those cut levels to provide a smooth, gouge free tool path.

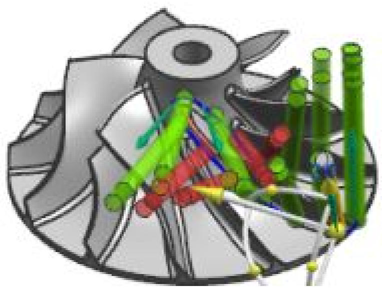
In the **Interpolated Vector** dialog box, you can:

- View a list of all control points.
- Change the tool orientation at system defined control points.
- Add, reorient, move, or delete user defined control points.
- Display a preview of tool orientations along the upper and lower rails.

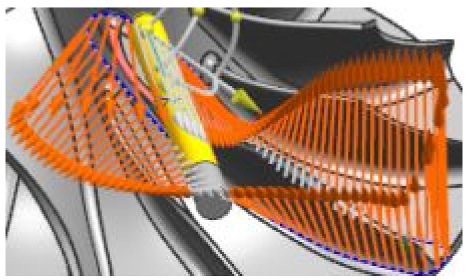
  By default, green tool positions are safe positions. Red tool positions are gouging positions.
- Dynamically rotate a vector and view the impact, including gouging, on the nearby and surrounding locations.
- Display the interpolated vectors along the upper and lower rails.

The **Interpolate Vector** tool axis option creates smooth gouge free tool paths. Use this option to overcome conditions where the **Automatic** tool axis option does not find gouge free orientation.

The **Interpolate Vector** option mainly controls the tool axis in the lead and lag directions. The processor also adds side tilt when it is required to avoid gouges.



Preview of tool orientations showing gouges



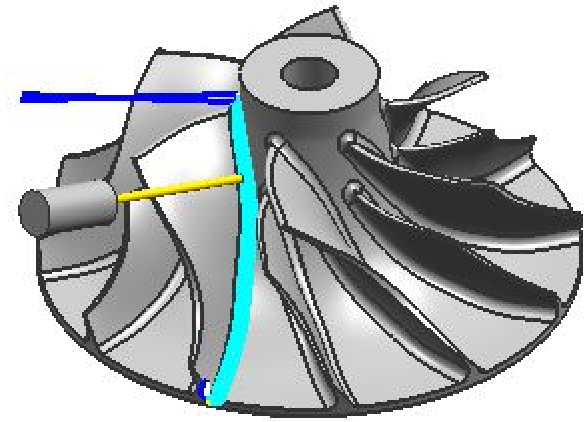
Preview of interpolated vectors

Where do I find it?

Application	Manufacturing
Prerequisite	Requires the *NX Turbo Machinery Add-on* license.
Toolbar	**Insert**® **Create Operation**
Menu	**Insert**® **Operation**
Location in dialog box	**Type** group® **mill_multi_blade**® **Operation Subtype** group® **MULTI_BLADE_ROUGH**, **HUB_FINISH**, **BLADE_FINISH**, or **BLEND_FINISH** [Operation] dialog box® **Tool Axis** group

Multi Blade Blend Finish operations

Use **Multi Blade Blend Finishing** operations to finish the blend areas of multiple blade impellers and blisks. You can finish the blade with a larger tool, and then finish the area between the blade and the hub with a smaller tool.



Where do I find it?

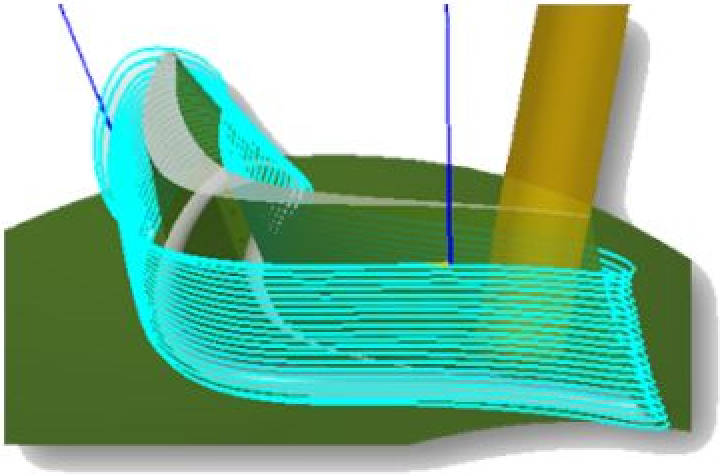
Application	Manufacturing
Prerequisite	Requires the *NX Turbomachinery Milling Add-on* license
Toolbar	**Insert**® **Create Operation** 
Menu	**Insert**® **Operation**
Location in dialog box	**Type** group® **mill_multi_blade**® **Operation Subtype** group® **BLEND_FINISH** 

Additional Multi Blade enhancements

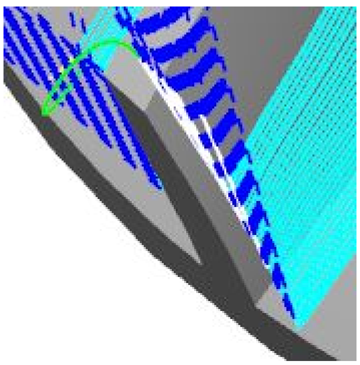
What is it?

You can add tangential extensions to the tool paths for **Blade Finish** and **Blend Finish** operations.

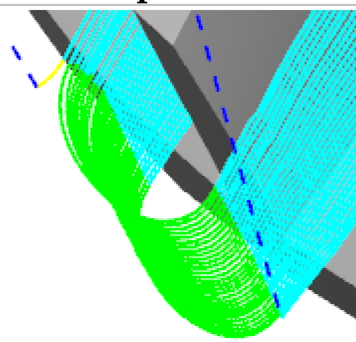
The **Blade Finish** operation includes the **Helical** cut pattern.



Blade Finish — all sides, helical cut pattern



Blade Finish — Left, Right, Leading Edge, without extensions



Blade Finish — Left, Right, Leading Edge, with tangential extensions

Where do I find it?

Application	Manufacturing
Prerequisite	Requires the *NX Turbomachinery Milling Add-on* license
Toolbar	**Insert® Create Operation**
Menu	**Insert® Operation**
Location in dialog box	**Type** group® **mill_multi_blade® Operation Subtype** group® **BLADE_FINISH** or **BLEND_FINISH**

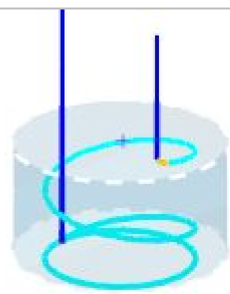
Hole Milling

Hole milling operations

What is it?

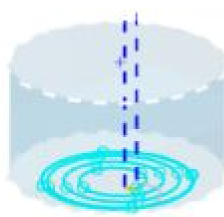
Use the new **HOLE_MILLING**  operation type to machine holes and cylindrical bosses with or without using Feature-based Machining. You can:

- Use all currently available tool types.
- Reverse the machining direction without affecting a feature’s orientation.
- Gouge check traverse moves.
- Machine a combination of through and blind holes.
- Extend the start and end of cuts.
- Specify helical and spiral cut patterns.
- Set multiple radial stepover passes.
- Set multiple axial stepovers for the **Spiral** and **Helical/Spiral** cut patterns.



The **Helical** cut pattern creates the following for a single axial level:

1. A helical cut motion.
2. An optional single turn cleanup pass at the depth of the hole or boss.



The **Spiral** cut pattern creates the following for each axial level:

1. A spiral cut motion.
2. An optional single turn cleanup pass at full diameter.

The **Helical/Spiral** cut pattern creates the following:

1. A single helical cut motion, in the center of the hole, to the axial cut depth.

2. A spiral cut motion with increasing diameter for each axial level.

3. An optional single turn cleanup pass at full diameter for each axial level.

Where do I find it?

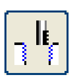
Application	Manufacturing
Toolbar	**Insert® Create Operation** 
Menu	**Insert® Operation**
Location in dialog box	**Type® hole_making/drill/planar_mill® Operation** **Subtype® HOLE_MILLING** 

Hole/Boss geometry object

What is it?

The **Hole/Boss**  geometry object helps you to define hole and boss locations with or without thread data. You can use this geometry object as a parent geometry object for **HOLE_MILLING** and **THREAD_MILLING** operations.

-  **HOLE_MILLING** operations support **Hole**, **Boss**, **Threaded Hole**, and **Threaded Boss** geometry.

-  **THREAD_MILLING** operations require thread information and only support **Threaded Hole**, and **Threaded Boss** geometry.

You can specify hole milling geometry by selecting any of the following:

- Points
- Curves
- Arcs
- Cylindrical edges
- Cylindrical faces

Why should I use it?

The **Hole/Boss** geometry object allows you to define holes and bosses with different tool axes, diameters, or depths.

Where do I find it?

Application	Manufacturing
Toolbar	**Insert**® **Create Geometry**
Menu	**Insert**® **Geometry**
Location in dialog box	**Type** group® **mill_planar/Drill/hole_making**® **Geometry** **Subtype** group® **HOLE_BOSS_GEOM**

IPW in Surface Contouring

What is it?

You can now use **Cutting Parameters** options in Surface Contouring operations to remove tool path segments that do not cut material and connect the remaining segments of the tool path. To recognize the blank and IPW from the previous operation, you must set the **In Process Workpiece** option to **Use 3D**.

You can also use other options to eliminate more cutting segments that do not remove material. The following options are now available for Surface Contouring:

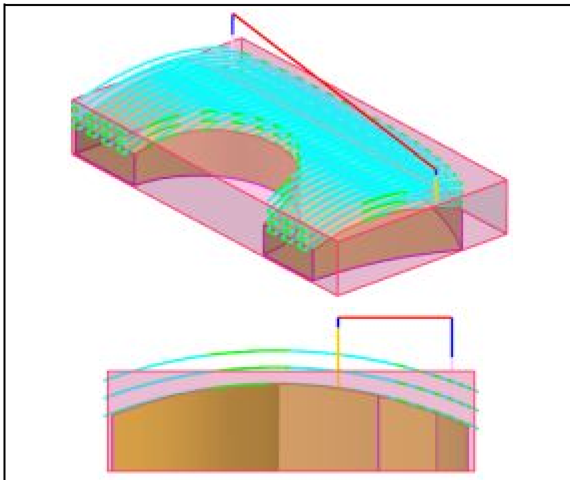
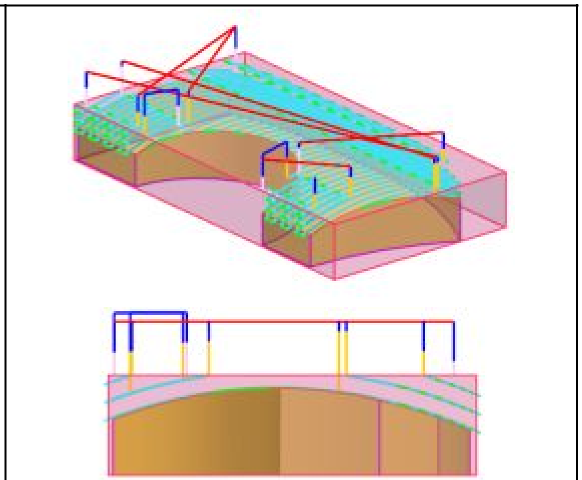
- **Hookup Distance**
- **Minimum Cut Length**
- **Minimum Material Removed**

The following options are new:

Apply to Last Pass — Controls whether blank sensitive trimming is applied on the last pass.

If the last pass finishes the part, clear this check box to ensure that the entire area is finished. Otherwise the finish pass is limited to only the regions where significant material remains on the IPW.

Minimize Non Cutting Moves — Optimizes the cutting order to reduce unwanted lifts

	
Before applying cutting parameters	After applying cutting parameters

For more information on the **Containment** tab options, see the Manufacturing Milling help.

Why should I use it?

Your machine time is reduced because NX eliminates segments of the tool path where there is no stock to remove.

Where do I find it?

Application	Manufacturing
Prerequisite	You must be in a valid fixed-axis surface contouring operation. You must use a **Zig**, **Zig-Zag**, or **Zig Zag with Lifts** cut pattern.
Location in dialog box	**Cutting Parameters** dialog box® **Containment** tab® **Blank** group® **In Process Workpiece** list® **Use 3D**

Customize Generic Motion and Probing operations

You can customize **Generic Motion** and **Probing** operations in the following ways:

- Reorder the list of sub-operation types that are available to create.
- Remove sub-operation types that you do not use from the list of available sub-operation types.
- Create custom sub-operations with the **User Defined Move** dialog item type.
- Add parameters to the standard **Probing** cycles and **Generic Motion** moves.

Note The following apply when you customize a **Probing** or **Generic Motion** dialog box:

- You cannot remove or modify the existing parameters in a standard sub-operation.
- You cannot modify a sub-operation that is currently used in the operation that you are customizing. You must delete any instances of the sub-operation before you can modify the sub-operation template.
- You can remove a sub-operation type that is currently used in the operation that you are customizing. The existing instances of the sub-operation remain in the operation.

Where do I find it?

Application	Manufacturing
Operation Navigator	Right-click a **Probing** or **Generic Motion** operation® **Object**® **Customize**
Dialog box	**Probing** or **Generic Motion** operation dialog box® **Options** group® **Customize Dialog**

Generic Motion enhancements

What is it?

The **Generic Motion** operation has the following enhancements:

- The **Rotary Move To Point** suboperation is now called **Rotary Point Vector Move**. This suboperation has better defaults for tool positioning. It also has new tool positioning options.

- There are two new suboperations: **Follow Curve/Edge** and **Follow Part Offset**.

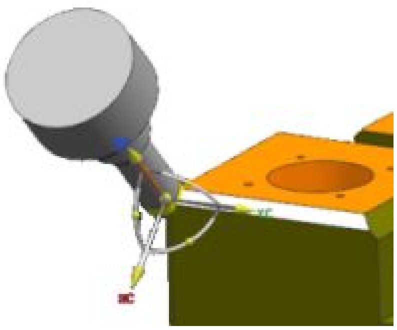
Rotary Point Vector Move

The **Rotary Point Vector Move** suboperation lets you dynamically drag the tool from its last position to a new position and orientation.

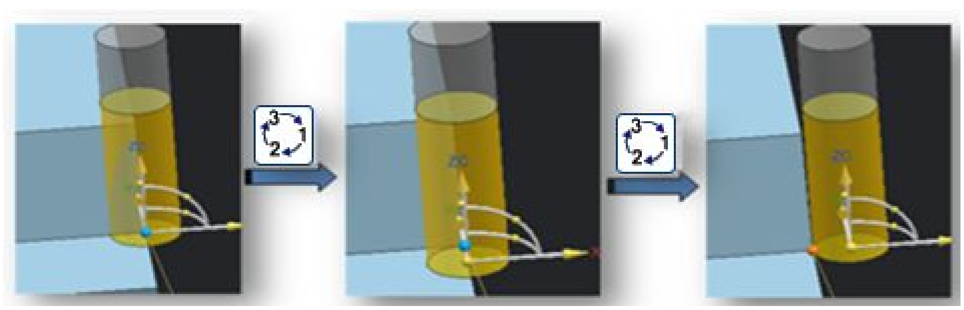
By default, the tool is positioned at one of two locations:

- If the tool position from a previous move suboperation is recognized by NX, the tool is positioned at that location.

- If the tool position from a previous move suboperation is not recognized by NX, then the tool is positioned at the MCS with the tool axis aligning with the ZM-axis.

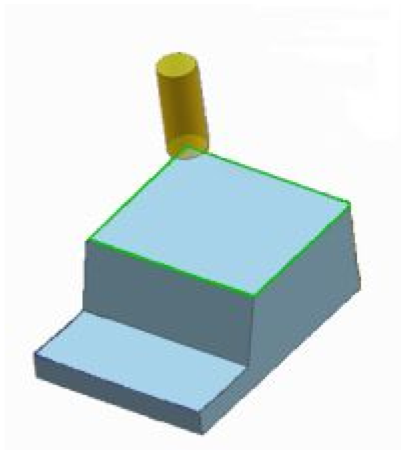


When the operation has geometry specified, you can position the tool tangential to a selected surface point. The new  **Alternate Solution** option selects between the *on* tool position and any *tangent* tool positions.



Follow Curve/Edge

The **Follow Curve/Edge** suboperation follows a series of directional curves.

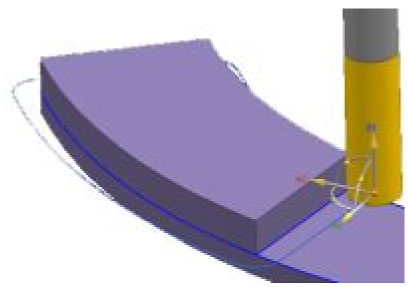


You can:

- Select curves or edges for the tool to follow.
- Specify start and end points and tool axes.
- Specify linear tangential extensions at the start and end points.

Follow Part Offset

The **Follow Part Offset** suboperation follows part cross sections.



You can:

- Specify start and end points.
- Specify the tool axis.
- Specify linear tangential extensions at the start and end points.

Why should I use it?

These enhancements make it easier for you to create tool paths that follow user specified tool positions when selecting points, tool axis, curves, and edges.

Where do I find it?

Application	Manufacturing
Toolbar	**Insert**® **Create Operation**
Location in dialog box	**Type** group® **Mill_Multi_Axis** or **Probing**® **Operation Subtype**® **GENERIC_MOTION**

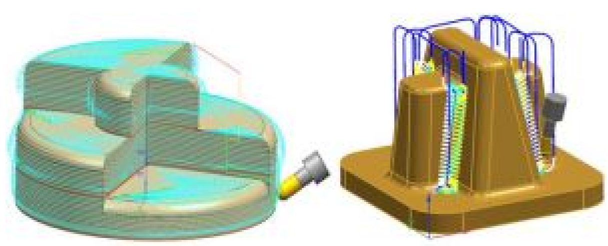
Tilt Tool Axis

Use the **Tilt Tool Axis** command to avoid tool holder collisions in Surface Contouring and ZLevel tool paths. The tool path is analyzed for tool holder collisions. If a collision is found, the tool axis is tilted to avoid the collision.

If you generate the operation after tilting the tool axis, the tilted tool path is replaced by the newly generated 3-axis tool path, and you need to repeat the tool axis tilt command.

Besides giving you the ability to avoid collisions, the **Tilt Tool Axis** command also allows tools to reach greater depths using the shortest tools possible for added rigidity.

In the examples, the tool axis tilts when a tool holder collision is detected, so that the tool can continue cutting down the wall towards the bottom of the part.



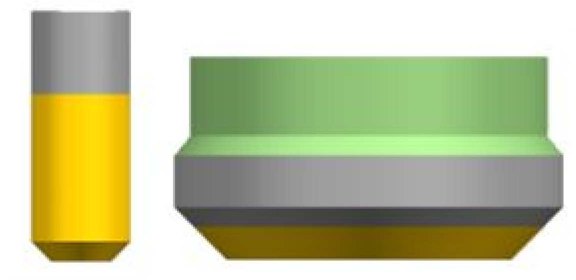
Where do I find it?

Application	Manufacturing
Prerequisites	• The operation must be a Surface Contouring or Zlevel operation. • The cutter must be a ball mill cutter with a tool holder defined.
Operation Navigator	Right-click an operation® **Tool Path**® **Tilt Tool Axis**

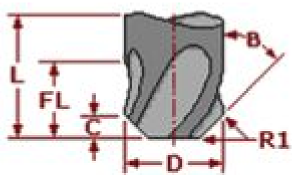
Chamfer mill tool type

What is it?

Use the new **Chamfer Mill**  tool type to create a mill tool that allows for an angular corner chamfer.



You can specify the following parameters for the tool:



(L) **Length**

(FL) **Flute Length**

(C) **Chamfer Length**

(D) **Diameter**

(R1) **Lower Radius**

(B) **Chamfer Angle**

Why should I use it?

This type of tool is useful when you need to define a face mill or chamfered end mill.

Where do I find it?

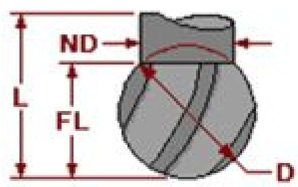
Application	Manufacturing
Menu	**Insert® Tool**
Toolbar	**Insert® Create Tool** 
Location in dialog box	**Type® mill_planar**, **mill_contour**, **mill_multi-axis**, or **mill_multi_blade® Tool Subtype® CHAMFER_MILL** 

Spherical mill tool type

What is it?

Use the new **Spherical Mill** tool type to create a ball-shaped tool with a reduced neck diameter. This type of tool allows you to machine restricted areas such as ports and undercuts.

You can define the shape of the tool using the following parameters:



(L) **Length**

(ND) **Neck Diameter**

(FL) **Flute Length**

(D) **Diameter**

Number of **Flutes**

Where do I find it?

Application	Manufacturing
Menu	**Insert® Tool**
Toolbar	**Insert® Create Tool**
Location in dialog box	**Type® mill_planar**, **mill_contour**, **mill_multi-axis**, or **mill_multi_blade® Tool Subtype® SPHERICAL_MILL**

Manufacturing Turning

Integrated test cut, probe, and finish cut

What is it?

You can create a single finish operation that incorporates the following sequence:

1. Test cut

2. Tool retracts to a safe position

3. Machine-specific probing cycle or pause for manual measurement

4. Finish cut

You can use the `OPSKIP` command to control whether the test cut and probing portion of the tool path is used or omitted.

You must create User Defined Events (UDEs) to manage the measuring operations.

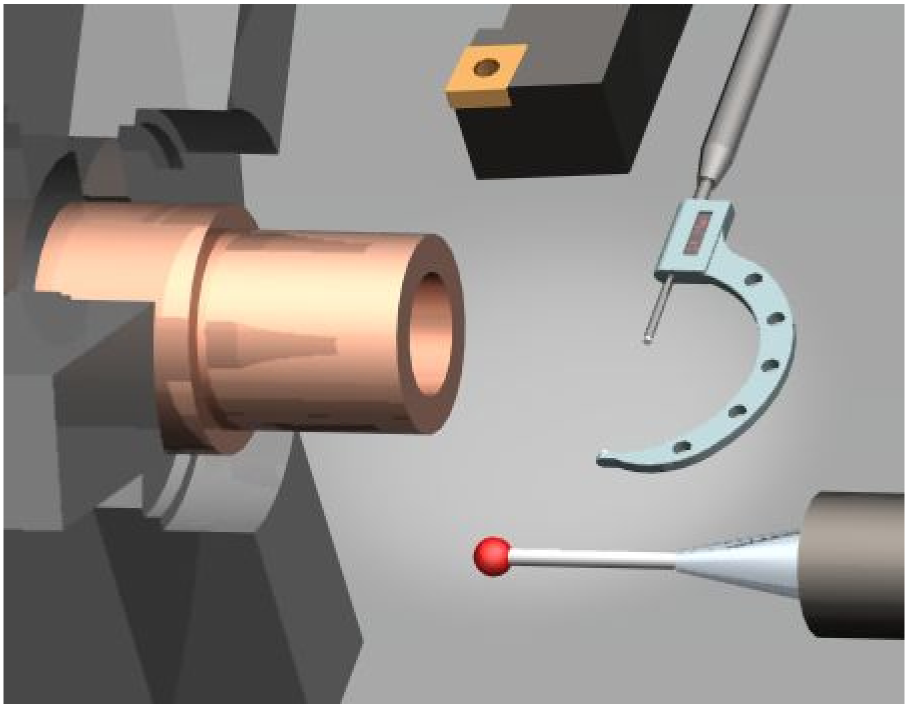
The following commands are available on the **Test Cut** tab of the **Cutting Parameters** dialog box.

- **Start of Test Cut Events**

- **Measuring Stop Events**

- **End of Test Cut Events**

- **Start of Finish Path Events**



The options are available in the **Cutting Parameters** dialog box, on the new **Test Cut** tab.

Where do I find it?

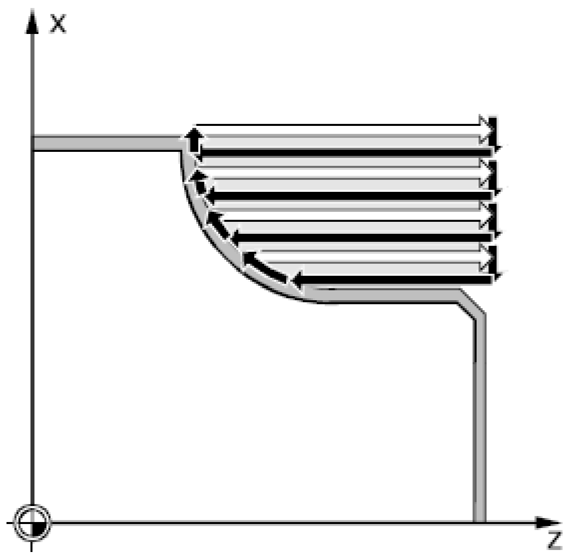
Application	Manufacturing
Prerequisite	Create or edit a turning operation You must have user defined events in place to control your probing attachment and communicate offset adjustments to the machine tool.
Toolbar	**Insert**® **Create Operation**
Menu	**Insert**® **Operation**
Location in dialog box	**Path Settings** group® **Cutting Parameters**® **Cutting Parameters** dialog box® **Test Cut** tab

Siemens Sinumerik 840D CYCLE95 Stock Removal

What is it?

The NX turning processor supports output for the Siemens Sinumeric 840d's `CYCLE95` stock removing cycle in the following operation types:

- FACING
- ROUGH_TURN_OD
- ROUGH_BACK_TURN
- ROUGH_BORE_ID
- ROUGH_BACK_BORE
- FINISH_TURN_OD
- FINISH_BORE_ID
- FINISH_BACK_BORE



Cycle parameters are automatically mapped from NX CAM turning parameters. Knowledge of specific output parameters is not required.

Note The IPW that is created by existing operations which are capable of CYCLE95 output is used by subsequent operations. This IPW will be only an approximation of the actual CYCLE95 results.

The Post Builder template post for the Siemens Sinumerik 840D Controller has been updated to include support for `CYCLE95`.

The contour definition for CYCLE95 is output in the form of a subprogram. You can output the subprogram in one of these ways:

- In-line with the main program
- At the end of main program
- In a separate SPF file

Why should I use it?

The post created by Post Builder using the Siemens 840D option now supports the Siemens controls optimally.

On the shop floor, the machine operator can adjust a single line of code, for example to change the cut depth per pass. To make the same change with conventional output, many lines of code must be recalculated.

Where do I find it?

Application	**NX**
Prerequisite	Create or edit a turning operation of a type listed in the preceding article.
Location in dialog box	**Machine Control® Motion Output® Machine Cycle**

Application	**Post Builder**
Prerequisite	Create or edit a **SIEMENS — SinumeriK_840D_lathe** post.
Location dialog box	**Program & Tool Path® Custom Command® PB_CMD_init_cycle95_output**

Manufacturing simulation and verification (ISV)

Controlling animation speed

What is it?

You can now control the animation speed in machine tool simulations by changing the number of frames that appear in a simulation. You can make a simulation run slower by increasing the number of animation frames, and run faster by decreasing the number of animation frames.

The **Minimum Simulation Time** option in the **Simulation Options** dialog box sets the amount of machining time that is required for one frame to be generated. For example, to increase the number of frames generated, decrease the **Minimum Simulation Time** value. This makes the animation take longer to play.

Note You may need to edit your machine tool definition file to make the **Minimum Simulation Time** option available, when the **Visualization** list is set to **Machine Code Simulate** in the **Animation** group in the **Simulation Control Panel** dialog box.

For the provided example:

```
…\MACH\samples\nc_simulation_samples\sim01_mill_3ax_cam_fanuc_in.prt,
```

you would edit the corresponding machine tool definition file,

```
…\MACH\resource\library\machine\installed_machines\sim01_mill_3ax\sim01_mill_3
```

insert a number sign (#) to comment out the second line.

```
MILL_3_AXIS,${UGII_CAM_LIBRARY_INSTALLED_MACHINES_DIR}sim01_mill_3ax\...
#CSE_FILES,${UGII_CAM_LIBRARY_INSTALLED_MACHINES_DIR}sim01_mill_3ax\...
```

Where do I find it?

Application	Manufacturing
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Manufacturing**® **Simulation & Visualization**® **ISV Options** tab® **Simulation Display**

IPW simulation options

What is it?

When you visualize material removal, you can now choose between motion-based IPW updates and length-based IPW.

Why should I use it?

Use motion-based rendering when you want the fastest performance in rendering the resulting IPW. The tool position may be updated several times during a motion, but the IPW is updated only after the motion is complete.

Use length-based rendering when you want to see the material removed in synchronization with the tool motion.

Where do I find it?

Application	Manufacturing
Graphics window	Right-click on an operation® **Tool Path**® **Simulate**
Location in dialog box	**Simulation Settings** group ® **Options**

Tool holder gouge checking

What is it?

You can now check for tool holder collision against part geometry when you simulate a tool path, using the **Check for Tool Holder Collision** option. In previous releases, you could only perform this check when you right-clicked an operation and chose **Tool Path**® **Gouge Check** from the **Operation Navigator**.

Why should I use it?

To find collisions between tool holder and design part during simulation.

Where do I find it?

Application	Manufacturing
Prerequisite	The part file requires a tool with a holder.
Operation Navigator	Right-click the operation® **Tool Path**® **Simulate**
Location in dialog box	**Simulations Options**® **Other Options** ® **Check for Tool Holder Collision**.

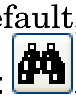
NX Post

NX Post enhancements

What is it?

In the **Postprocess** dialog box, you can now change the options for **Output Warning Messages** and **Activate Review Tool** specified with a post in **Post Builder**. The default settings are **Post Defined**. You can set the options to **On** or **Off**. The settings you select persist during your NX session.

You can control whether your users see the review tool options by changing the **Show Review Tool options** customer default.

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

Tolerance resolutions and distance calculations have been improved for better performance.

Why should I use it?

The improved user interface makes it easier to set the options for **Output Warning Messages** and **Activate Review Tool**.

Where do I find it?

Application	Manufacturing
Operation Navigator	Right-click the selected operation® **Post Process**
Location in dialog box	**Settings** group

Standard Sinumerik cycles in supplied machines

What is it?

All installed machines under the *${UGII_BASE_DIR}\MACH\resource\library\machine* folder now have standard Sinumerik cycles that do the following:

- Add Multi spindle, multi-channel mill-turn examples.
- Add machine tool examples that allow machines to change tool heads.
- Allow standard NX simulation drivers to support PLANE SPATILA for the Heidenhain TNC controller and CYCLE800 for the Sinumerik controller.

Why should I use it?

The standard Sinumerik cycles support the creation of mill turn posts.

Post Builder

Dual units posts in Post Builder

What is it?

You can enable your main post to function as a dual-unit post by using the **Units Only Subpost** option. When you select this option and specify a main post, Post Builder automatically detects the units of the main post and creates a subpost to output the alternate units.

For example, if your main post is *fanuc_3ax*, which is in inches, and you select the **Units Only Subpost** option, Post Builder creates the *fanuc_3ax__MM* subpost. If this subpost exists in the same folder as the *fanuc_3ax* post, and you ask for millimeter output in NX, the postprocessor automatically finds and uses the *fanuc_3ax__MM* subpost.

Note If your fanuc_3ax post in is millimeters, then the subpost name would be fanuc_3ax__IN. There are two underscores in the subpost name before the uppercase IN or MM.

When you create a new main post, you can use the following options in the **Optional Alternate Units Subpost** group.

Default Accepts the default name for a future units-only subpost. Your new main post will search for this name in the folder if you specify opposite units in NX. You must create the units-only subpost.

Specify Lets you specify an existing units-only subpost. The file must exist, and should have similar parameters to your new main post.

Why should I use it?

The **Units Only Subpost** option makes it easy for you to develop posts that are capable of properly processing output in both inches and millimeters.

Where do I find it?

Creating a new units-only subpost

Application	Post Builder
Windows	**Start**® **All Programs**® **Siemens NX 8.0**® **Manufacturing Tools**® **Post Builder**
Location in dialog box	**File**® **New**® **Units Only Subpost**

Identify an existing units-only subpost

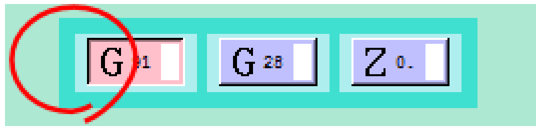
Application	Post Builder
Windows	**Start**® **All Programs**® **Siemens NX 8.0**® **Manufacturing Tools**® **Post Builder**
Location in dialog box	**Output Settings**® **Other Options** tab® **Optional Alternate Units Subpost** group

Post output conditions

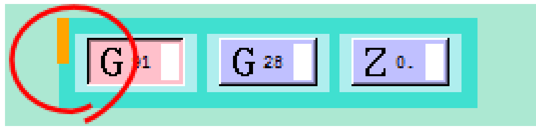
What is it?

You can create, edit, or remove custom output conditions for a block or macro from Post Builder by using the **Output Condition** command. For example, you can write Tcl code to enable or suppress output under specified conditions, or to force the block output into two lines of code. In previous releases you could only do this by editing the Tcl program.

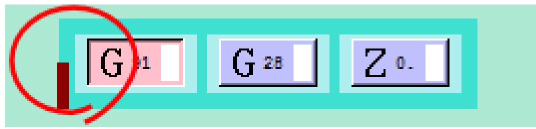
You can also prevent the code sequence numbers from appearing in the output for a block or macro by using the **Suppress Sequence Number** command.



Default appearance of a block

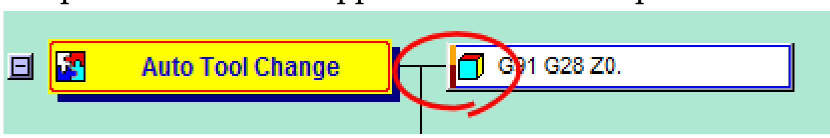


Block with sequence numbers suppressed



Block with an output condition

Block with sequence numbers suppressed and an output condition



Appearance on the **Program** tab, where you can create, edit, or remove conditions.

Why should I use it?

You can easily find the lines of code for output conditions in Post Builder, instead of searching through a large Tcl program.

Where do I find it?

Application	Post Builder
Windows	**Start**® **All Programs**® **Siemens NX 8.0**® **Manufacturing Tools**® **PostBuilder**
Location in dialog box	**Program & Tool Path** tab® **Program** tab® right-click a block or macro® **Set Output Conditions** or **Suppress Sequence Number**

User Defined Events enhancements

What is it?

Your post can now inherit user defined events from one or more other posts.

- When you select the **Inherit UDE From a Post** check box, you can enter the names of the posts whose user defined events you want the current post to inherit.

- When you select the **Include Own CDL File** check box, you can now select a folder containing *.cdl* files whose user defined events you want the current post to include.

If your referenced files contain events with the same names and different versions or revisions of the event, the order in which you specify posts or *.cdl* files is important. Same-named events in the last file to be read are the version that is used. The software reads the lists starting with the first entry, and ending with the last entry.

Why should I use it?

You can reuse the code for existing user defined events instead of creating the code again.

Where do I find it?

Application	**PostBuilder**
Windows	**Start**® **All Programs**® **Siemens NX 8.0**® **Manufacturing Tools**® **PostBuilder**
Location in dialog box	**NC Data Definitions** tab® **Other Data Elements** tab® **User Defined Events** group® **Include Other CDF File** or **Inherit UDE From a Post**

Suboperation events

What is it?

A new **Miscellaneous** page under **Tool Path** allows you to specify events for the start and end of suboperations in generic motion or probing operations. The following events are generated automatically in NX:

- `MOM_start_of_subop_path`

- `MOM_end_of_subop_path`

Why should I use it?

These new event markers make it convenient for you to configure suboperation event handlers in Post Builder.

Where do I find it?

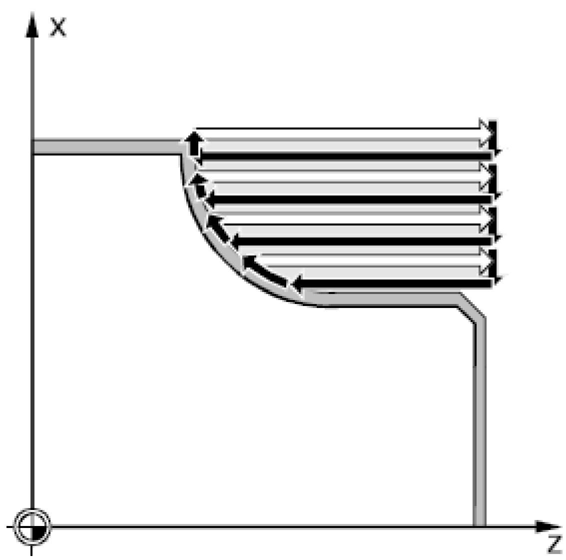
Application	**PostBuilder**
Windows	**Start**® **All Programs**® **Siemens NX 8.0**® **Manufacturing Tools**® **PostBuilder**
Location in dialog box	**Program & Tool Path** tab® **Program** tab® **Tool Path** group® **Miscellaneous** page® **Start of Subop Path** and **End of Subop Path**

Siemens Sinumerik 840D CYCLE95 Stock Removal

What is it?

The NX turning processor supports output for the Siemens Sinumeric 840d's `CYCLE95` stock removing cycle in the following operation types:

- FACING
- ROUGH_TURN_OD
- ROUGH_BACK_TURN
- ROUGH_BORE_ID
- ROUGH_BACK_BORE
- FINISH_TURN_OD
- FINISH_BORE_ID
- FINISH_BACK_BORE



Cycle parameters are automatically mapped from NX CAM turning parameters. Knowledge of specific output parameters is not required.

Note The IPW that is created by existing operations which are capable of CYCLE95 output is used by subsequent operations. This IPW will be only an approximation of the actual CYCLE95 results.

The Post Builder template post for the Siemens Sinumerik 840D Controller has been updated to include support for `CYCLE95`.

The contour definition for CYCLE95 is output in the form of a subprogram. You can output the subprogram in one of these ways:

- In-line with the main program
- At the end of main program
- In a separate SPF file

Why should I use it?

The post created by Post Builder using the Siemens 840D option now supports the Siemens controls optimally.

On the shop floor, the machine operator can adjust a single line of code, for example to change the cut depth per pass. To make the same change with conventional output, many lines of code must be recalculated.

Where do I find it?

Application	**NX**
Prerequisite	Create or edit a turning operation of a type listed in the preceding article.
Location in dialog box	**Machine Control® Motion Output® Machine Cycle**

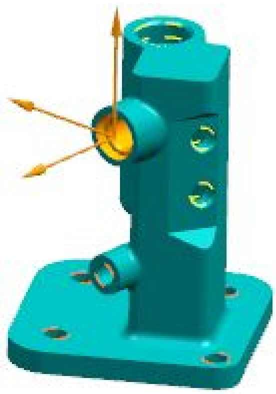
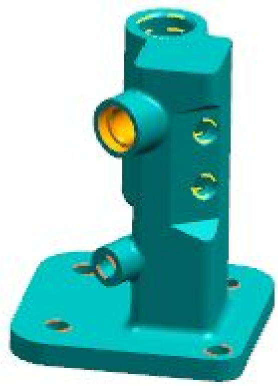
Application	**Post Builder**
Prerequisite	Create or edit a **SIEMENS — SinumeriK_840D_lathe** post.
Location dialog box	**Program & Tool Path® Custom Command® PB_CMD_init_cycle95_output**

Feature-based machining

Show Feature CSYS

What is it?

Use the **Show Feature CSYS** option to turn on or off the coordinate system that displays the position and orientation of machining features.

 **Show Feature CSYS** on	 **Show Feature CSYS** off

Why should I use it?

This command helps you to control the direction display for machining features without using a customer default.

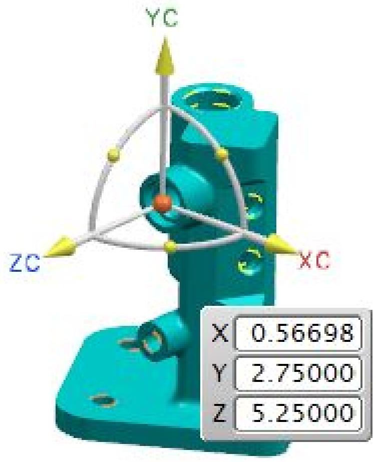
Where do I find it?

Application	Manufacturing
Machining Feature Navigator	Right-click in the background® **Show Feature CSYS**

Edit Feature CSYS

What is it?

Use the new **Edit Feature CSYS** command to edit the position and orientation of a feature. You can change the CSYS settings either dynamically in the graphics window or in the **Edit Feature CSYS** dialog box.



In the **Details** panel of the **Machining Feature Navigator**, the **Overriden** column displays a check mark for the modified CSYS attributes.

Attribute	Value	Overriden	Original Value
Y_ORIENTATION_L	–0.8660	✔	0.0000
Y_POSITION	2.7500		2.7500

Where do I find it?

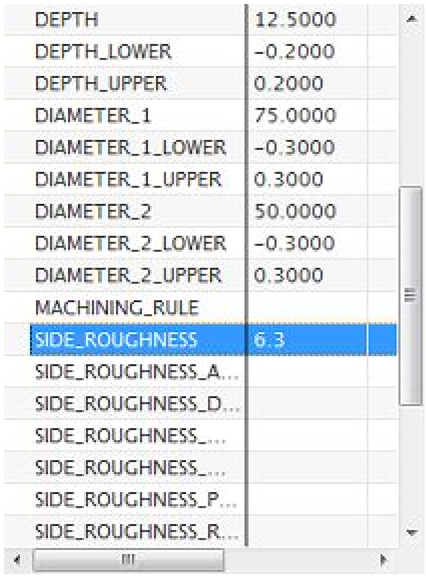
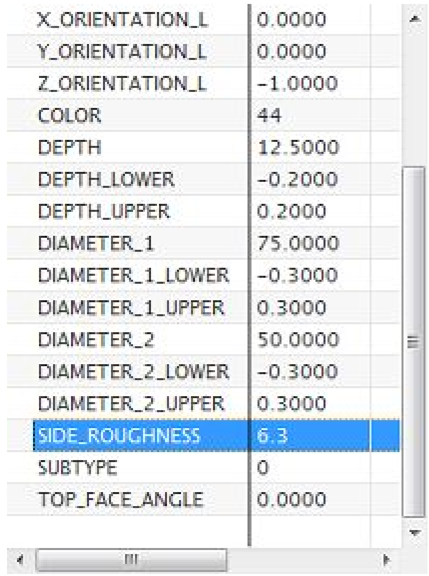
Application	Manufacturing
Machining Feature Navigator	Right-click the selected feature® **Edit Feature CSYS**

Hide Undefined Attributes

What is it?

Use this option to hide in the **Details** panel of the **Machining Feature Navigator**, those machining feature attributes that have no defined values.

Parametric recognition adds empty attributes when there are no values to extract from the model file. For example, there are multiple attributes that remain empty when side roughness is not completely defined.

Hide Undefined Attributes off	**Hide Undefined Attributes** on
	

Why should I use it?

Hide undefined attributes so that you can more easily locate the attributes that you want.

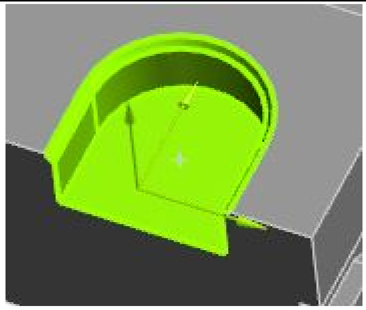
Where do I find it?

Application	Manufacturing
Machining Feature Navigator, **Details** panel	Right-click in the background® **Hide Undefined Attributes**

Teach Features

Use the **Teach Features** command to define parametric recognition rules for your new custom feature types. After you select the feature geometry in the part, the command automatically generates the recognition code and stores the results in the knowledge library. New feature types are available for reuse immediately after they are created.

Because you can define your own feature types without manually writing the feature recognition code, the time needed to create your NC program is significantly reduced. If you want to automatically generate new machining rules from existing operations for the new features, use the **Teach Machining Rules** command.



New feature type

Where do I find it?

Application	Manufacturing
Prerequisite	Requires the *NX30435 — NX Feature Based Machining Author* license.
Menu	**Tools® Machining Feature Navigator® Teach Features**

Teach Machining Rules

Use the **Teach Machining Rule** command to automatically generate new machining rules from existing operations within Manufacturing. The new machining rules are stored in a machining knowledge library so that the Machining Knowledge Editor (MKE) can access them. The following operation parameters are supported:

- Feature parameters
- Tool parameters
- Cutting parameters
- Non-cutting move parameters
- User defined events
- Cycle parameters
- Geometry parent
- KF parameters

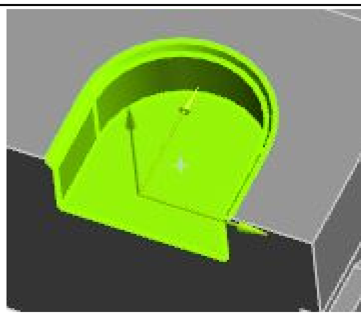
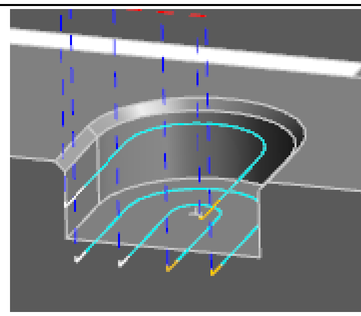
The **Teach Machining Rule** command improves efficiency.

- Because you can generate and test the operations for a feature before you generate the machining rules, you can speed up the process for defining new machining rules.

- Because you can define your own machining rules without manually writing the rule definition code, you can reduce the time needed to create your NC program. It can be time consuming to set the right operation parameters in a machining rule and define the necessary **Rule Add-on** items.

  To make the rules more general and applicable to features with different dimensions, it is recommended that you edit the rules in the MKE.

Without editing the machining rules, you can use the **Create Feature Process** command to create the same set of operations for features with the same dimensions. After editing the machining rules, you can create the same set of operations for features with different dimensions.

	
New feature type	Operations to teach

Where do I find it?

Application	Manufacturing
Prerequisites	• The *NX30435 — NX Feature Based Machining Author* license. • An instance of the feature for which the rules will be applicable. • Operations defined for the feature. • A machining knowledge library to store the rules.
Operation Navigator	Right-click the feature operations® **Object**® **Teach Operations**

Chapter

6 CAE (Digital Simulation)

NX 8 Advanced Simulation

Solver version support

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 8 releases

Solver	File Type	NX 8
NX Nastran	Import ASCII (.dat)	8
	Import Binary (.op2)	8
	Export ASCII (.dat)	8
	Post-processing of Results (.op2)	8
MSC Nastran	Import ASCII (.dat)	2011.1
	Import Binary (.op2)	2011.1
	Export ASCII (.dat)	2011.1
	Post-processing of Results (.op2)	2011.1
Abaqus	Import ASCII (.inp)	6.10
	Import Binary	N/A
	Export ASCII (.inp)	6.10
	Post-processing of Results (.fil)	6.11
	Post-processing of Results (.odb)	6.10-EF1
ANSYS	Import ASCII (PREP7, CDB)	13
	Import Binary (.rst, .rth)	13
	Export ASCII (.inp)	13
	Post-processing of Results	13
LS-DYNA	Import ASCII	971R5.0
	Import Binary	N/A
	Export ASCII (.k)	971R5.0
	Post-processing of Results	971R5.0

NX7 releases

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2	NX 7.5.3
NX Nastran	Import ASCII (.dat)	6.1	7.0	7.0	7.1	7.1
	Import Binary (.op2)	6.1	7.0	7.0	7.1	7.1
	Export ASCII (.dat)	6.1	7.0	7.0	7.1	7.1
	Post-processing of Results	6.1	7.0	7.1	7.1	7.1
MSC Nastran	Import ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010
	Import Binary (.op2)	2008r1	2008r1	2008r1	2008r1	2010
	Export ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010
	Post-processing of Results	2008r1	2008r1	2008r1	2008r1	2010

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2	NX 7.5.3
Abaqus	Import ASCII (.inp)	6.8-1	6.9–1	6.9–1	6.9-1	6.10
	Import Binary	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9	6.9	6.9	6.10
	Post-processing of Results (.fil)	6.8-EF2	6.9.2	6.9.2	6.10-1	6.10-1
	Post-processing of Results (.odb)	6.8-EF2	6.9-EF1	6.9-EF2	6.9-EF2	6.10-EF1
ANSYS	Import ASCII (PREP7, CDB)	12	12.1	12.1	12.1	13
	Import Binary (.rst, .rth)	12	12.1	12.1	12.1	13
	Export ASCII (.inp)	12	12.1	12.1	12.1	13
	Post-processing of Results	12	12.1	12.1	12.1	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A
	Import Binary	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
	Post-processing of Results	N/A	N/A	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

Advanced Simulation Help improvements

This release contains a number of enhancements to the Advanced Simulation Help.

Advanced Simulation help reorganization and improvements

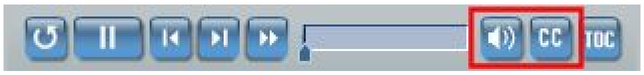
The Advanced Simulation Help has been reorganized to make navigation more logical and relevant information easier to find.

- Help for all Advanced Simulation dialog boxes is now available in a new Command reference section. Dialog boxes are listed in alphabetical order.
- Help topics that cover the process of modeling contacting or glued surfaces are now located in a new Contact and Glue Conditions section.
- Each solver environment now has a separate section in the Advanced Simulation Help.

CAE videos with audio

The video examples available in the Advanced Simulation Help now include voice-over narration. This narration provides important information about the steps being demonstrated as well as additional details about complex tasks.

For each video, you can use the options on the play bar to display closed caption text or to mute the narration.



To access the CAE video examples, see Video examples.

NX Open documentation for CAE

This release includes new NX Open documentation specifically for CAE.

- It includes information about the NX Advanced Simulation solver language and the NX Advanced Simulation mesh properties

- It supplements the complete API definitions found in the NX Open Reference manuals.

For more information, see **Open CAE documentation overview**.

General capabilities

Browsing to results files

What is it?

Use the new **Browse** command to open the directory containing the files generated by the solver, including the input and results files.

Why should I use it?

This command is useful if you need to review the solver-generated files or examine them to troubleshoot problems with the solution.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file
Simulation Navigator	Right-click the solution container® **Browse**

New default file names

What is it?

In this release, when you create a new FEM, Simulation, or Assembly FEM file from the **Simulation Navigator**, the default file name has changed. The new file name uses the master part file name as a prefix, as follows:

- The default FEM file name uses the format *CADpartname_fem(number).fem*. For example, a FEM file created from the CAD part *adjust_arm.prt* has the default name of *adjust_arm_fem1.fem*.

- The default Simulation file name uses the format *FEMpartname_sim(number).sim*, because the FEM is the master part for the Simulation file. For example, a Simulation file created from the FEM file *adjust_arm_fem1.fem* has the default name of *adjust_arm_fem1_sim1.sim*.

- The default assembly FEM file name uses the format *CADpartname_assyfem(number).afm*. For example, an assembly FEM file created from the CAD part *adjust_arm.prt* has the default name of *adjust_arm_assyfem1.afm*.

When you click **New** from the **Standard** toolbar or choose **File**® **New**, the default file names do not use the master part file name as a prefix, because the master part has not yet been defined.

To modify the default name of the files that you create from the **Simulation Navigator**, select **Prefix Part Name with Master Part Name** on the **Customer Defaults** dialog box. If you deselect this option, the file names are the customer defaults defined for the **File New** dialog box.

Where can I find it?

Creating a new FEM, Simulation, or Assembly FEM file from the **Simulation Navigator**:

Application	Advanced Simulation
Simulation Navigator	Right-click the part or ipart name® **New FEM** Right-click the part or ipart name® **New FEM and Simulation** Right-click the part name® **New Assembly FEM** Right-click the FEM name® **New Simulation**

Setting a different default file name:

Application	Advanced Simulation
Menu	**File**® **Utilities**® **Customer Defaults**® **Simulation** category® **General** subcategory® **FE Model and Simulation Create** tab® **Prefix Part Name with Master Part Name**

Update Agent

What is it?

You can use the **Add Update Agent** command to identify particular solutions within a simulation to re-solve automatically when the solid geometry or the points, curves, or coordinate systems in the master CAD part are modified. The mesh, boundary conditions, and solutions can be updated.

Part modifications that you make while the FEM, idealized part, or Simulation file are displayed do not trigger a re-solve.

This feature works with the FEM object in the Product Template Studio application.

For steps, see Set up an Update Agent for use with Product Template Studio.

Where do I find it?

Update Agent customer defaults

Menu	**File**® **Utilities**® **Customer Defaults**® **Simulation**® **General**
Location in dialog box	**Allow UpdateAgent Creation**

Update Agent command

Simulation Navigator	Right-click the solution® **Add Update Agent** Right-click the solution® **Remove Update Agent**

Persistent element quality displays

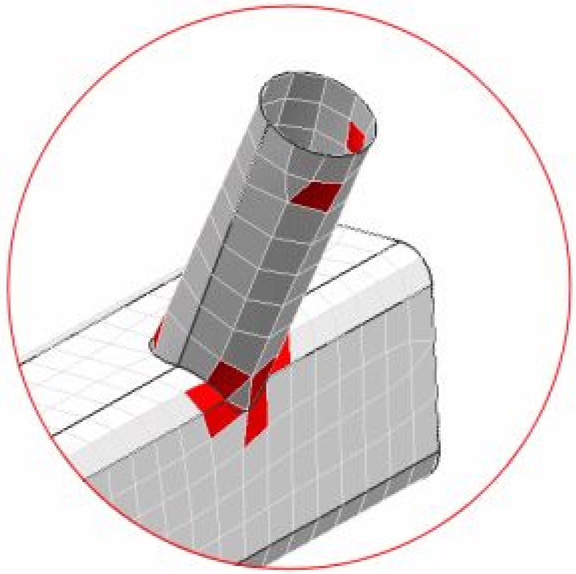
What is it?

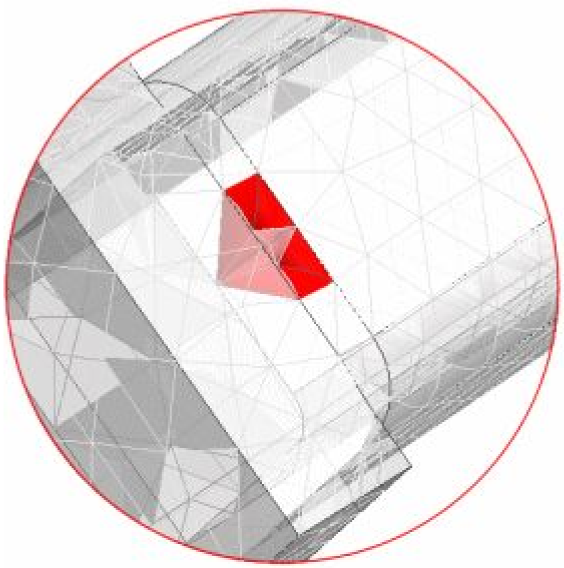
In this release, the **Model Display** preference includes new options for displaying elements that fail element shape quality checks.

In previous releases, when you wanted to identify problems in your mesh, you used the **Model Checks** command to check element shapes. When you use this command, the problem elements are displayed until you choose another command. In this release, in addition to using **Model Checks**, you can also generate a persistent display of the problem elements. To do this, use **Model Display** and set the **Color Basis** option to **Element Quality Checks**.

The element quality model display options let you control colors for elements that pass or fail quality checks. You can also use translucency to display elements. The **Threshold Values** option opens the **Threshold Values** dialog box, where you can modify the values for the element shape checks.

The figure shows two element quality displays. The top image shows 2D elements, and the bottom image shows 3D elements with translucency.





Where do I find it?

Application	Advanced Simulation
Menu	**Preferences® Model Display**

Units conversion for scalar values

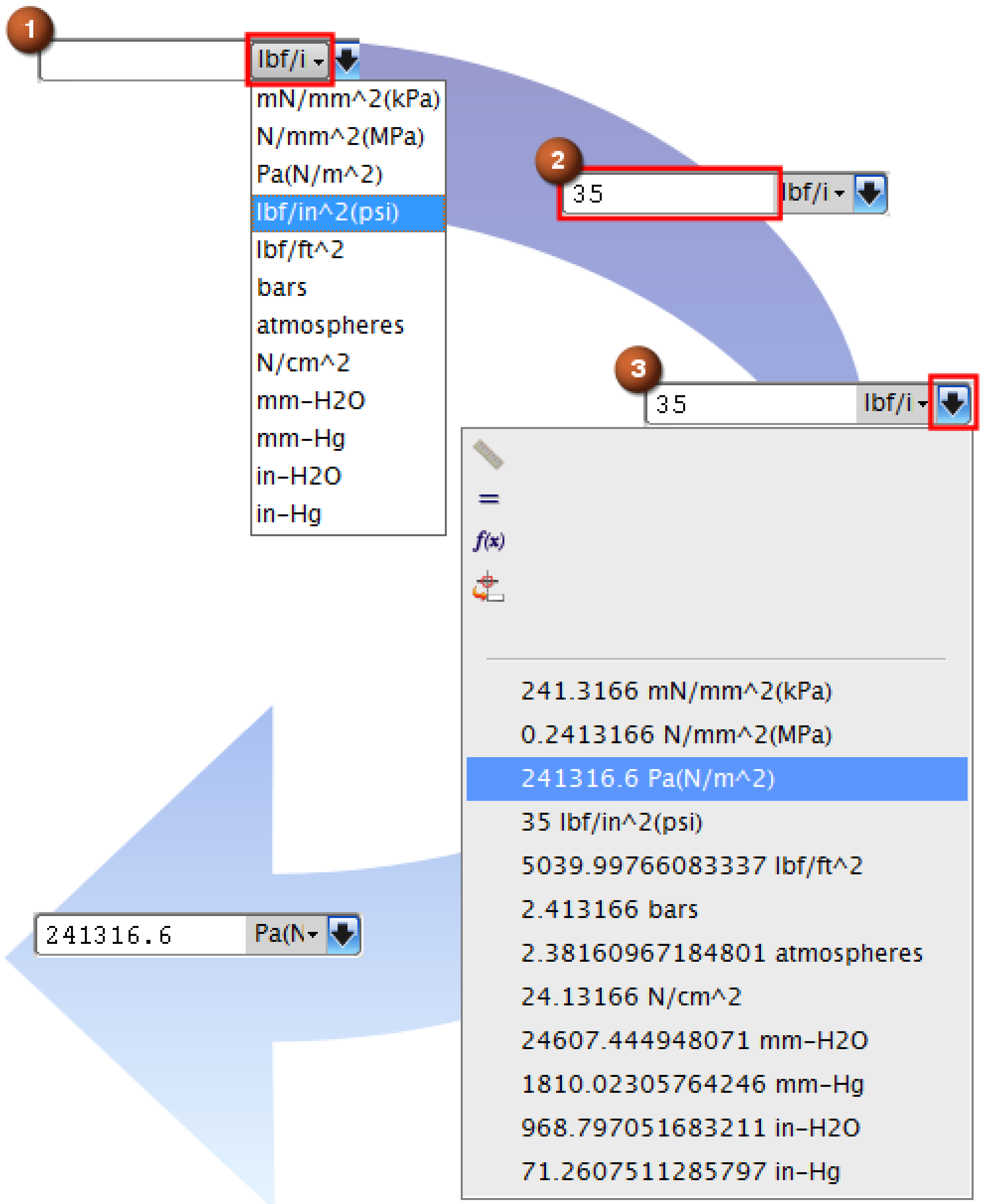
What is it?

When you enter scalar values to define parameters for your model, the **DesignLogic** list now provides automatic units conversion. When you specify units and enter a value, click  to choose a conversion value.

This option is available throughout Advanced Simulation whenever you enter a scalar value with units in a dialog box, including but not limited to:

- Material property tables
- Physical property tables
- Global and local element lengths
- Element- and mesh-associated data
- Loads, constraints, and simulation objects

The following example shows how to convert pressure entered in pounds per square inch to pascals.



(1) Select the input units. (2) Enter a value. (3) Select the conversion value.

Why should I use it?

Use this feature to convert input parameters into consistent model units. This feature is especially useful when specifications that define your analysis problem, such as material data or loading conditions, are not provided in units consistent with your model.

Where do I find it?

Application	Advanced Simulation
Location in dialog box	**DesignLogic** list

Polygon geometry and geometry abstraction

Suppressing holes in sheet bodies

What is it?

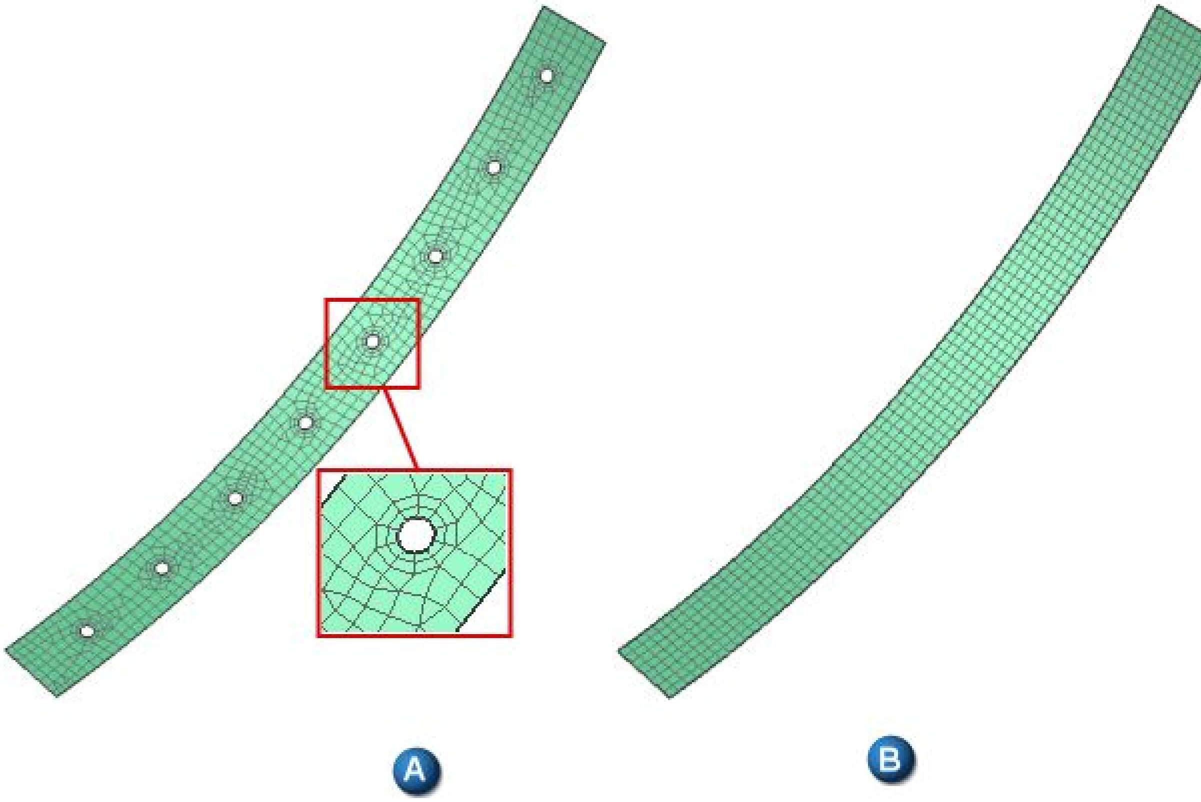
Some models may contain holes that have little impact on its overall stiffness. In analyses in which you are interested in the overall behavior of a structure and not the stresses in the vicinity of the holes, you may want to ignore selected holes to eliminate unnecessary mesh density. You can use the new **Suppress Hole** command to remove holes from sheet bodies, such as midsurface sheet bodies. You can remove:

- Selected holes or all holes whose diameter is smaller than a specified threshold value.
- Holes contained within a single face.
- Holes that span multiple faces.
- Both circular and non-circular holes.

When the software removes the hole, you can choose to create either a point or a mesh point at the centroid of the hole. You may want to create a mesh point at the hole's centroid, for example, so you can later create an FE-based connection element, such as an RBE2, at that location.

You can use the **Suppress Hole** command either before or after you create a mesh on the part. If you use **Suppress Hole** to remove holes from a face that you have already meshed, you must use the **Update Finite Element Model** command to update the mesh.

The following graphic shows an example of how you can use **Suppress Hole** to automatically remove a number of holes at one time. (A) shows a 2D mesh on a midsurface sheet body that contains a series of 4 mm holes. (B) shows the updated mesh on the sheet body after the **Suppress Hole** command was used to remove all holes with a diameter smaller than 5 mm.

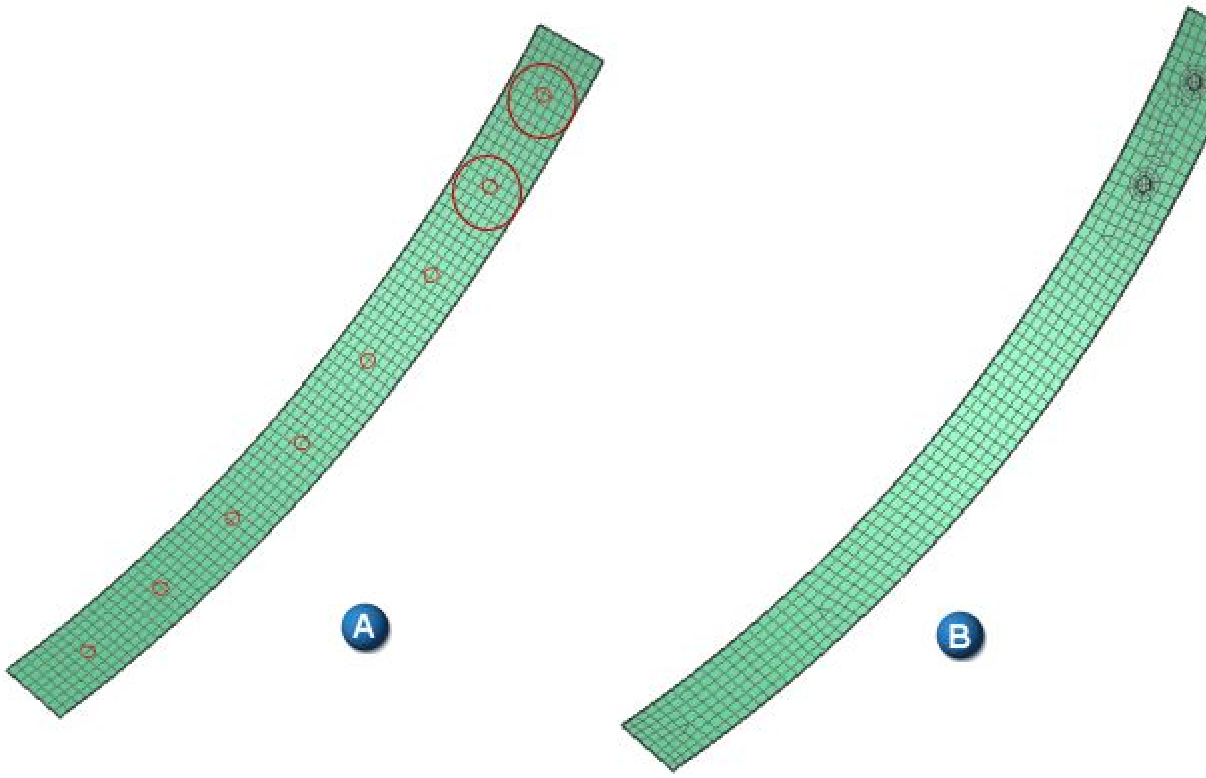


For more information, see Suppressing holes in sheet bodies.

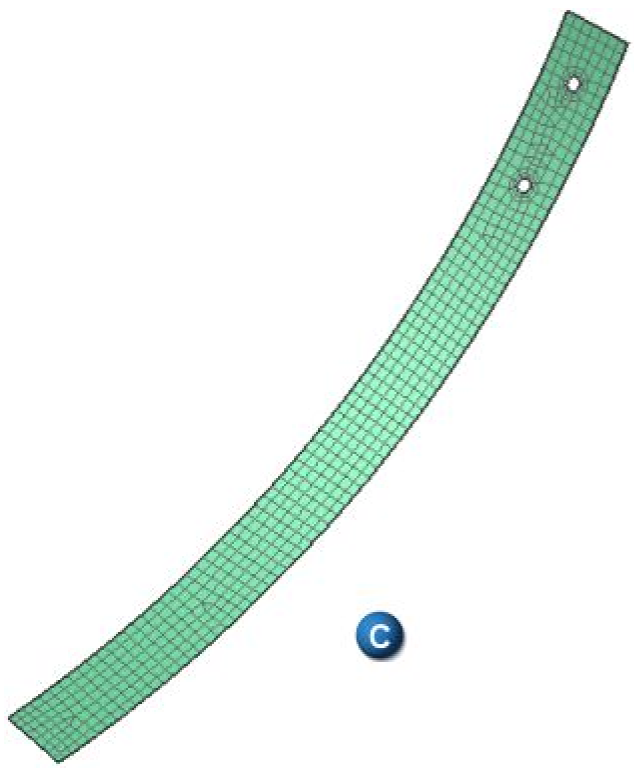
Unsuppressing holes

After you suppress a hole, you can use other **Model Cleanup** commands to unsuppress that hole. You first use the **Split Face** command with the **Split face by suppressed edges** option to unsuppress the edge that defines the perimeter of the hole. You then can use the **Repair Face** command to remove the face inside the perimeter of the hole.

The following graphic shows an example of unsuppressing holes. (A) shows the edges that are currently suppressed. The **Split Face** command with the **Split face by suppressed edges** option was to unsuppress the two highlighted edges. (B) shows the updated mesh with the unsuppressed edges.



(C) shows the updated mesh after the **Repair Face** command was used to remove the face on the inside of each hole's perimeter.



Replaying automatic hole suppression after geometry updates

The software stores information about suppressed holes in the **CAE Geom Recipes** node in the **Simulation Navigator**. If the CAD geometry in either the master or the idealized part is updated, the software uses the recipe to try to re-suppress the holes when the FEM file is updated.

CAE Geom Recipes

1 – holes in belt top side
2 – holes in belt bottom side

Note The software can only store information about suppressed holes in the **Simulation Navigator** if the CAD part associated with your FEM file is loaded at the time you use the **Suppress Hole** command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file active
Toolbar	**Advanced Simulation® Suppress Hole**
Menu	**Model Cleanup® Suppress Hole**

Tools for identifying and repairing polygon body problems

What is it?

When you create a FEM file, if you select the **Associate to Part** option in the **New FEM** dialog box, the software automatically creates polygon geometry from the geometry in your CAD model. As you work with that polygon geometry, polygon bodies may develop data consistency or corruption problems. In other cases, you may make unintended modifications to that polygon geometry.

This release includes new tools that you can use to:

- Identify polygon bodies that have inconsistent or corrupt data and repair those inconsistencies.
- Recover from unintended modifications to the polygon geometry.

Identifying problematic polygon bodies

This release includes a new **CAE Model Consistency** option in the **Model Check** dialog box. You can use this option to check all polygon bodies in your FEM file for issues. If the software identifies problems, it lists the affected polygon bodies by name and stores the modified bodies in an output group. As a best practice, you should use the **CAE Model Consistency** option periodically while you are building your finite element model.

Note Before you use this option, you must have the CAD part that is associated with the current FEM file loaded.

Repairing polygon bodies

If the software identifies polygon bodies with consistency issues, and the CAD part that is associated with the current FEM file is loaded, you can use new options in the **Simulation Navigator** to fix those issues.

When you right-click a polygon body in the **Simulation Navigator**, there are three new options.

- The **Recreate and Update** option deletes the selected polygon body and creates a new polygon body using the original CAD geometry. The software then updates the body so that any mesh mating conditions, meshes, loads, boundary conditions, or CAE geometry recipes are preserved and updated. Use this option when you want to recreate the problematic polygon body and preserve as much of the data that was associated with that body as possible.

- The **Recreate Only** option deletes the selected polygon body and creates a new polygon body using the original CAD geometry. The software also deletes any:

  - Mesh mating conditions.

  - Modifications made with commands on the **Model Cleanup** toolbar.

  - Meshes.

  - Loads or boundary conditions associated with the deleted polygon body.

- The **Delete** option removes the selected polygon body from your FEM file. You may want to use the **Delete** option to exclude a particular body from your analysis.

  When you delete a polygon body from your model, the software moves that body into a new **Excluded Polygon Geometry** folder in the **Simulation Navigator**.

 Excluded Polygon Geometry

 Polygon Body

If you later decide to include the body in your analysis, right-click the body in the **Excluded Polygon Geometry** folder and select **Create Polygon Body** to create a new polygon body.

Note Although you can use the **Delete** option to delete polygon bodies that were created by the **Face from Mesh** command, you cannot use the **Recreate and Update** or **Recreate Only** options on those bodies.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file active
Simulation Navigator	Right-click an existing polygon body and select **Recreate and Update**, **Recreate Only**, or **Delete**

Advanced control for polygon body tessellation

What is it?

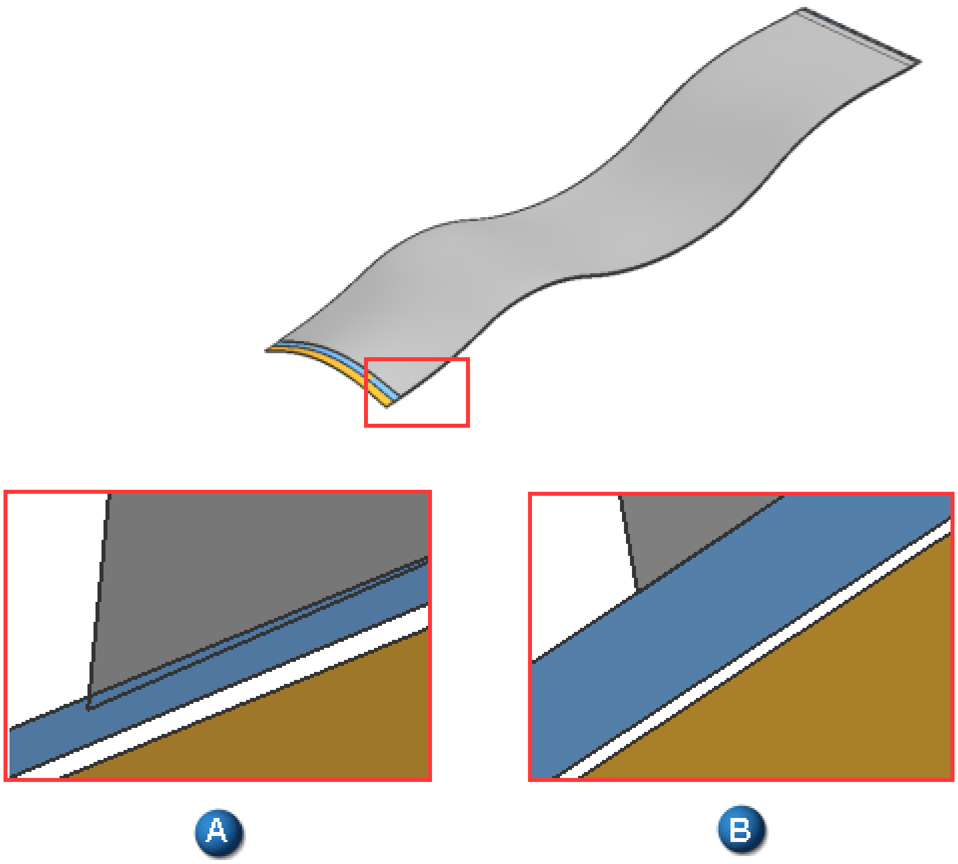
When you create a FEM file, you can use the new **High Resolution Polygon Bodies** option to have the software use a finer level of tessellation to create the edges in the polygon bodies.

- To turn this capability on in your current NX session, use the **High Resolution Polygon Bodies** option in the **Meshing Preferences** dialog box.

- To turn this capability on by default, use the **High Resolution Polygon Bodies** customer default.

Using a finer level of tessellation is helpful when you want to ensure that software accurately represents the curvature of surfaces in a very thin body. In some cases, using the standard resolution for polygon bodies on thin, curved (though nearly flat) surfaces from different bodies can cause problems when you try to create **Mesh Mating Conditions** between them.

This graphic shows the subtle differences between polygon geometry that was created with standard resolution (A) and polygon geometry that was created with the **High Resolution Polygon Bodies** option turned on (B). In (B), notice how the overlap between the blue and the gray faces has been eliminated and how the gap between the blue and the yellow faces is smaller.



Because the **High Resolution Polygon Bodies** option increases the number of facets in a polygon body, the size of the associated FEM file also increases.

Where do I find it?

Meshing Preferences option

Application	Advanced Simulation
Menu	**Preferences® Meshing**

Customer default

Application	Advanced Simulation
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Simulation® Meshing® General** tab

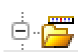
Measure Bodies improvements for polygon bodies

What is it?

This release includes improvements to the **Measure Bodies** command for polygon bodies. In previous releases, the **Measure Bodies** listing was not accurate or complete for polygon bodies.

- In the **Measurement Mass Properties** section of the listing, the listed values are now correct for polygon bodies.
  - If the selected polygon bodies have an assigned material, **Measure Bodies** calculates their weight and mass based on the specified density for the material.
  - If the selected polygon bodies do not have an assigned material, the software uses a density of 1.0 in the current units to calculate the mass properties, such as volume and area.
- In the **Detailed Mass Properties** section of the listing, the software now calculates all properties for polygon bodies. These properties include, for example, radii of gyration, moments and products of inertia, as well as principal moments. In previous releases, when you used **Measure Bodies** to analyze polygon bodies, the software did not calculate any of the properties listed in the **Detailed Mass Properties** section.

Additionally, if you select the **Associative** option in the **Measure Bodies** dialog box, the software creates a **Measures** feature in the associated FEM file. You can view this feature from the **Part Navigator**:

 Measures

 Body Measurement

Where do I find it?

Application	Advanced Simulation
Menu	**Analysis® Measure Bodies**

Meshing

Enhancements to Nastran spot weld connections

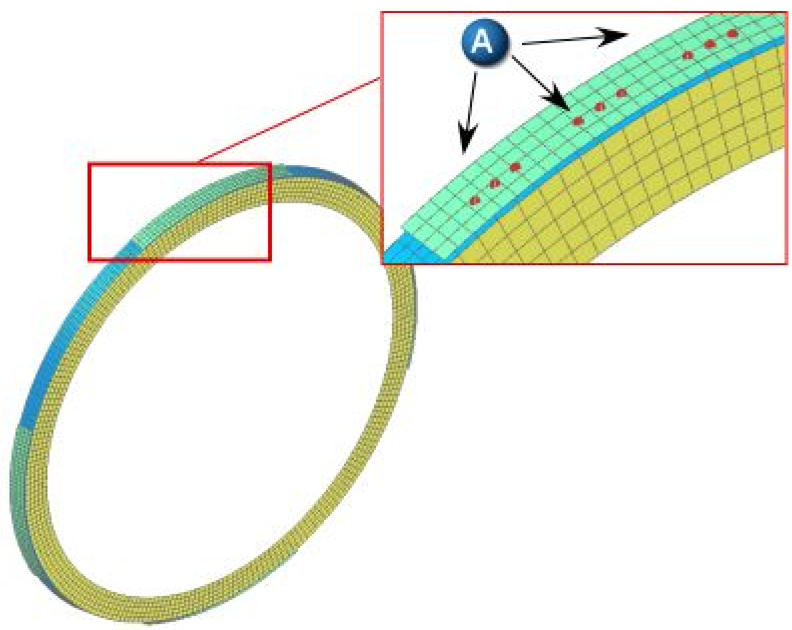
What is it?

When you select the **Point on Curve/Edge** option in the **CFAST/CWELD Connection** dialog box, you can now create a series of CFAST or CWELD connections along an edge or curve. In this release, you can create a series of CFAST or CWELD connections:

- That are located along an edge or curve or are offset from a selected edge or curve.
- In a specified pattern that repeats along the length of the edge or curve.

These improvements allow you to more easily replicate spot weld or fastener point patterns from the manufacturing process in your CAE model.

The graphic below shows an example of the new CFAST and CWELD connection capabilities. The model is comprised of several individual sheet bodies that will be welded together during the manufacturing process. (A) shows the locations of the CWELD connections that are offset from one edge on the top body. Notice that the CWELD connections were created in a pattern, with one inch between each individual connection and three inches between each group of connections.



For more information, see CWELD and CFAST Connections (Nastran).

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® CFAST/CWELD Connection** 
Menu	**Insert® Mesh® CFAST/CWELD Connection**

Ability to project nodes directly onto CAD geometry

What is it?

When you create a 2D mesh, you can now have the software project nodes directly onto the underlying CAD geometry rather than onto the CAE polygon geometry.

- To turn this capability on in your current NX session, use the new **Project Nodes to CAD Geometry** option in the **Meshing Preferences** dialog box.

- To turn this capability on by default, use the new **Project Nodes to CAD Geometry** customer default.

The **Project Nodes to CAD Geometry** option is most useful when you need to create a mesh for a contact analysis and want to ensure the accuracy of node locations in regions of contact.

Note If you select the **Project Nodes to CAD Geometry** option, you must have the idealized part loaded before you can generate a mesh.

In previous releases, you could use the **Node Proximity to CAD Geometry** and **Adjust Node Proximity** options in the **Model Check** dialog box to evaluate and modify the proximity of nodes relative to the CAD geometry. However, you could only use this option after you had already meshed the model.

Where do I find it?

Meshing Preferences option

Application	Advanced Simulation
Menu	**Preferences® Meshing**

Customer default

Application	Advanced Simulation
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Simulation**® **Meshing**® **General** tab

Beam section reuse in managed mode

What is it?

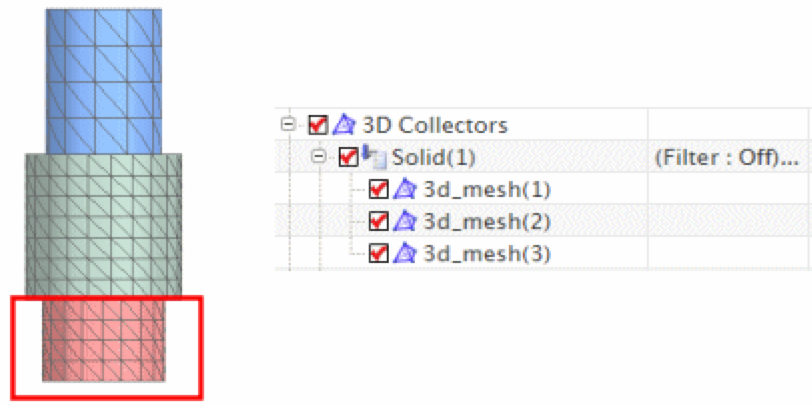
NX 7.5.2 introduced the capability to define a standard, General Geometry, or Face of Solid type of beam cross section and save it as a template in the NX Reuse Library. This capability is now supported in managed mode. For more information, see **Unsatisfied xref title**.

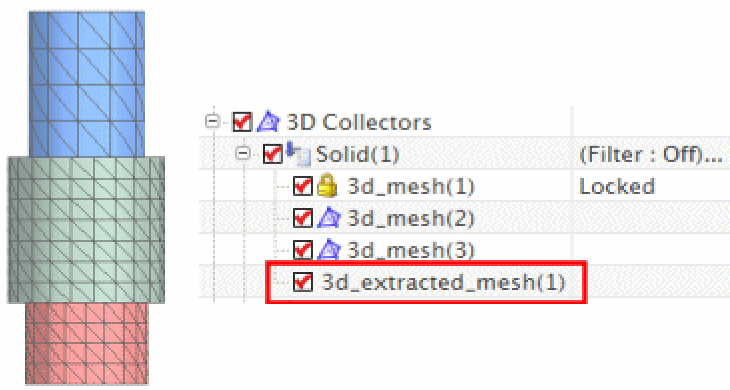
Element Extract enhancements

What is it?

Beginning with this release, when you use the **Element Extract** command, the extracted mesh retains the mesh associated data and display properties of the source mesh. In previous releases, the extracted mesh used default settings for mesh associated data and display properties.

The graphic shows the meshes listed in the **Simulation Navigator** before and after using **Element Extract**. The source mesh includes all of the pink elements in **3d_mesh(1)**. The new mesh is **3d_extracted_mesh(1)**.





Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file with meshes displayed.
Toolbar	**Element Operations** toolbar® **Element Extract**
Menu	**Edit**® **Element**® **Extract**

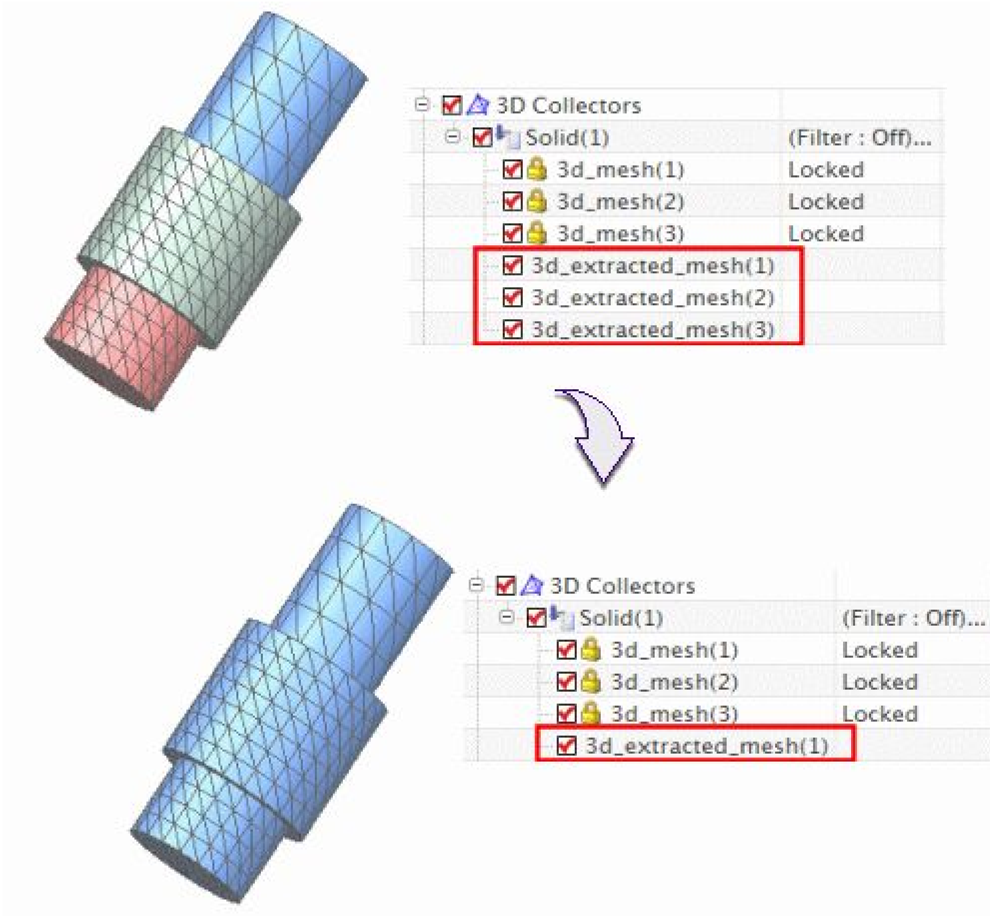
Merge Meshes enhancements

What is it?

Beginning with this release, when you use the **Merge Meshes** command, the first mesh that you select becomes the merged mesh. The software adds the remaining selected meshes to the first mesh. The merged mesh retains characteristics of the first mesh, including the mesh name, mesh associated data, and mesh display settings.

In previous releases, merged meshes were placed in a new **Simulation Navigator** node that had a unique name. The merged mesh display used default settings.

The graphic shows the mesh names and display colors before and after three extracted meshes are merged.



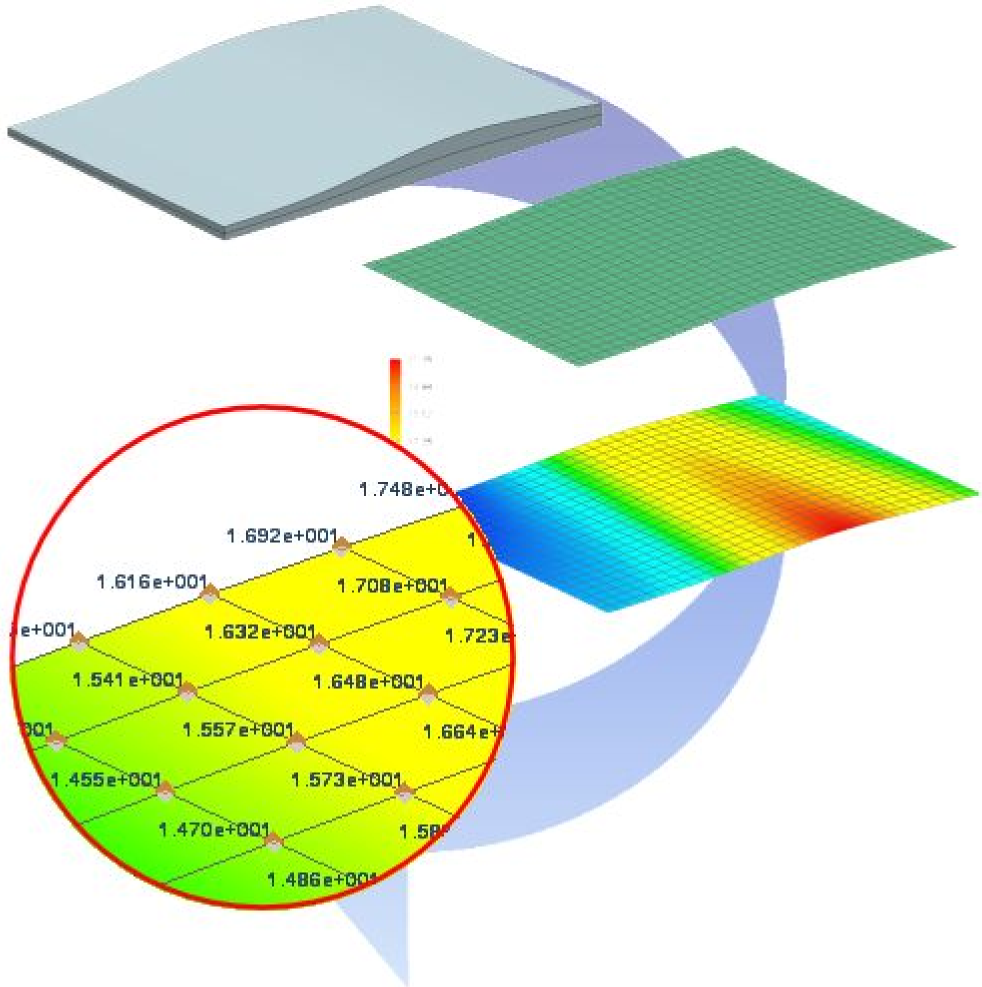
Where do I find it?

Application	Advanced Simulation
Prerequisite	Meshes that are not associated with geometry and are located in the same mesh collector.
Simulation Navigator	Select multiple meshes® right-click a selected mesh® **Merge Meshes**

Shell element thickness contour displays

What is it?

Use the **Plot Thickness Contours** command to generate a contour plot of shell element thicknesses as a standard post view. When you generate a thickness plot, the **Post-Processing** toolbar is enabled, and you can use the commands on this toolbar to view and interrogate element thicknesses.



Clockwise from top: A midsurface on a thin, sculpted part. A 2D mesh on the midsurface. A thickness contour plot. Thickness values extracted using Identify Results.

Why should I use it?

You can use **Plot Thickness Contours** to verify your mesh, to generate high-quality visualizations for reports or presentations, and to interrogate and extract thickness data.

You can use **Edit Post View** to modify the thickness display, and you can use **Identify Results** to probe thickness values at nodes and write thickness information to a spreadsheet or comma-separated-value file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file containing one or more 2D meshes.
Simulation Navigator	Right-click a 2D mesh collector node ® **Plot Thickness Contours** Right-click a 2D mesh node ® **Plot Thickness Contours**

New meshing customer defaults

What is it?

This release includes several new meshing customer defaults.

Default available for pyramid element transitions

You can use the new **Transition with Pyramid Elements** customer default to control the default setting of the **Transition with Pyramid Elements** option in the **3D Tetrahedral Mesh** dialog box. In the NX Nastran and ANSYS solver environments, you use the **Transition with Pyramid Elements** option to create pyramid elements to transition between a tetrahedral mesh and an adjacent hexahedral mesh.

Default available to allow minor modifications to 2D seed meshes during tetrahedral meshing

This release also includes a new **Allow Changes to 2D Seed Meshes During Tetrahedral Meshing** customer default that allows the software to make adjustments to existing 2D seed meshes during tetrahedral meshing.

In previous releases, when you created a 3D mesh on a body that already had an existing 2D mesh on one or more faces, the software could only split edges to facilitate mesh generation on cylinders. This meant that during tetrahedral meshing, the software could not merge any sliver faces or portions of faces into a face with an existing 2D mesh.

When you set the **Allow Changes to 2D Seed Meshes During Tetrahedral Meshing** customer default, the software can perform a greater level of abstraction during 3D meshing.

- If you select this customer default, the software will eliminate sliver faces or the sliver portions of previously meshed faces to facilitate a higher quality tetrahedral mesh.

- If you do not select this customer default, the software will not merge sliver faces. However, as in previous releases, the software will continue to split the edges in existing 2D meshes to facilitate tetrahedral meshes through cylinders.

Note If you select the new default, you can use the **Lock** command in the **Simulation Navigator** to prevent the software from modifying selected 2D meshes.

Where do I find it?

Application	Advanced Simulation
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Simulation® Meshing® General** tab

Ability to create multiple 0D elements

What is it?

In the **Element Create** dialog box, when you select **0D** from the **Element Family** list, you can now use **Smart Selection** options to quickly create a number of 0D elements. For example, you can:

- Use the **Related Nodes** option to create 0D elements on all nodes related to a selected polygon face.

- Use the **Nodes by Group** option to create 0D elements on all nodes in an existing group that you select from the **Simulation Navigator**.

In previous releases, you had to select locations for 0D element creation one at a time.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® Element Create** 
Menu	**Insert→Element→Create**

Improved handling of sheet bodies for 2D meshing

What is it?

When you use the **Polygon Body** option in the selection filter to create a 2D mesh, the software now stores information about the associated CAD sheet body with your mesh definition. If you make modifications to the associated CAD sheet body, such as adding a new face, the software automatically adds that new face into the 2D mesh.

Note If you use the **Polygon Face** option in the selection filter to create the 2D mesh, the software does not store information about the CAD sheet body with the mesh definition.

In previous releases, the software only stored information about the faces from the sheet body and not about the sheet body itself. Modifications that you made to the associated CAD sheet body were not reflected in the mesh.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 2D Mesh** 
Menu	**Insert® Mesh® 2D Mesh**

More comprehensive information for meshes

What is it?

When you use the **Information** command to list information about a selected mesh, the software generates a more comprehensive listing of information for geometry-based meshes. The information report now includes a new **Mesh Recipe** section, which lists all the options you selected or defined when you created the mesh. For example, for a mesh you created with the **2D Mesh** command, the software lists the element type and specified size as well as the settings for options including **Match Edges** and **Export Mesh to Solver**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Simulation Navigator	Right-click an existing mesh and select **Information**
Menu	**Information® Advanced Simulation® Mesh**

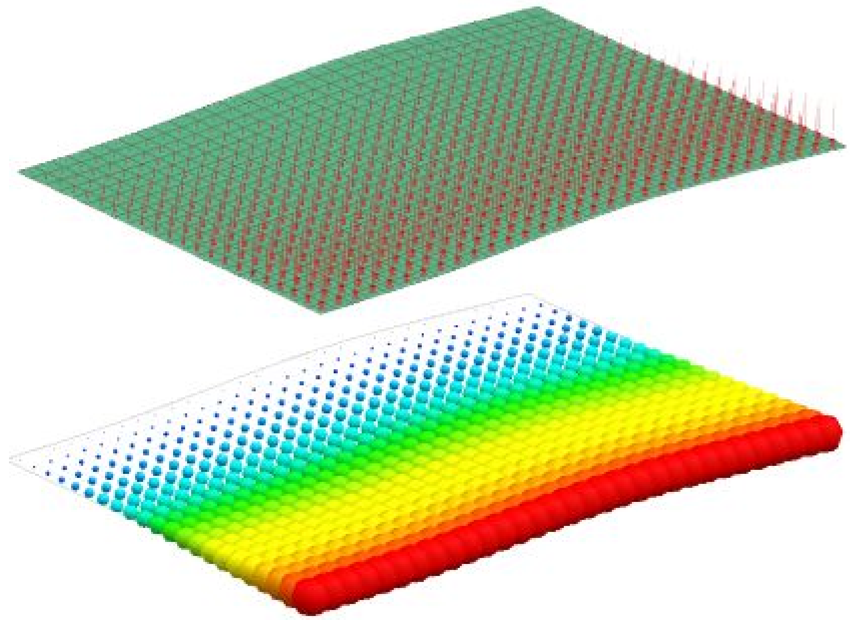
Boundary conditions

Pressure and temperature contour displays

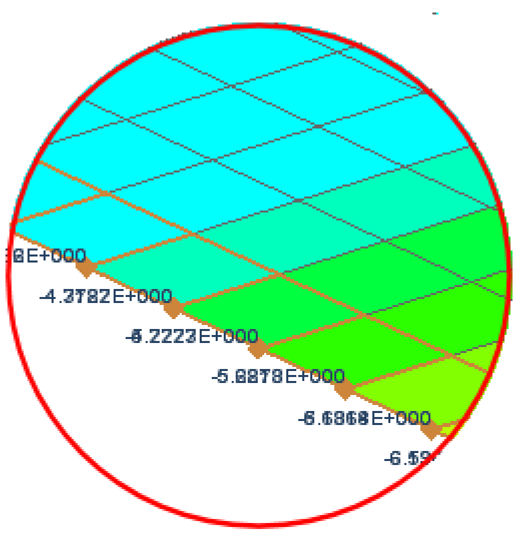
What is it?

Use the **Plot Contour** command to generate a contour plot of pressure, nodal pressure, or temperature loading as a standard post view.

When you generate a contour plot of pressure or temperature, the **Post-Processing** toolbar is enabled, and you can use these the commands on this toolbar to view and interrogate your loading conditions.



Top: A parametric spatially varying pressure load. Bottom: The same load as a spherical marker plot.



Pressure values on an edge extracted using the Identify Results command.

Why should I use it?

You can use the **Plot Contour** command to verify your loading conditions, to generate high-quality visualizations for reports or presentations, and to interrogate and extract loading data.

You can use the **Edit Post View** command to modify the loading display, or you can use the **Identify Results** to probe loading values at nodes and write them to a spreadsheet or comma-separated-value (CSV) file.

For time-varying pressure or temperature, you can plot the loading conditions at any arbitrary point in time.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file containing one or more temperature or pressure loads.
Simulation Navigator	Right-click a pressure, nodal pressure, or temperature load node ® **Plot Contour**

Axial deformation loads for 1D elements

What is it?

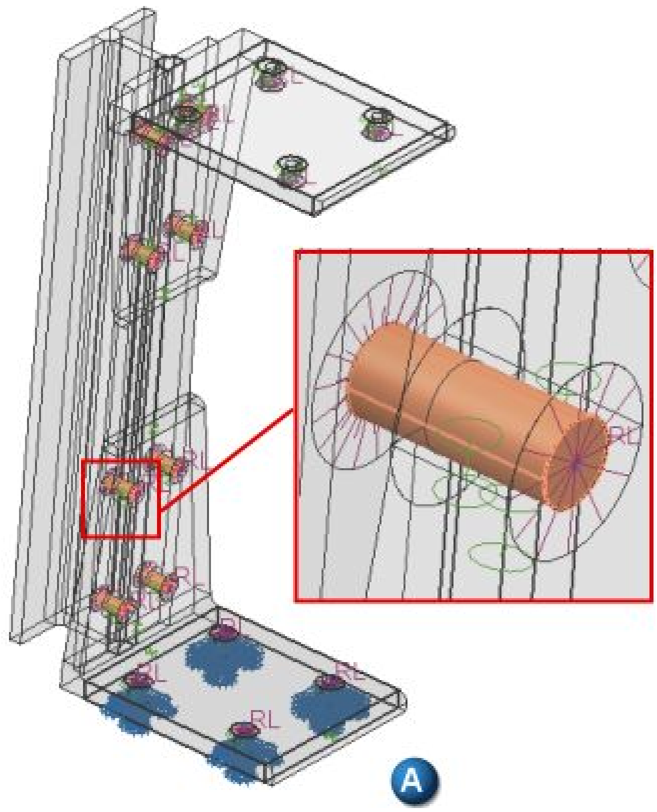
In the Nastran environment, you can now use the new **1D Axial Deformation** command to apply an enforced axial deformation (deflection) to CBAR, CBEAM, CONROD, CROD, and CTUBE type elements.

In NX, you can use the **1D Axial Deformation** command to apply axial deflections in Solution 101, 105, and 200 type analyses. Enforced deflections are useful, for example, when you analyze misfit or misalignment conditions in structures.

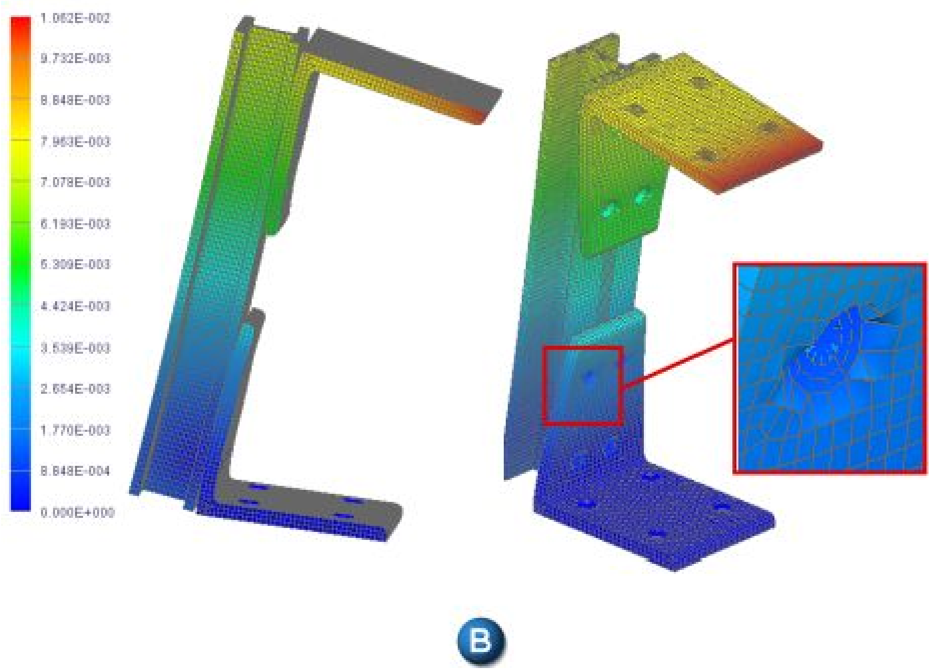
The **1D Axial Deformation** command does not enforce a strain or apply an extensional length to the 1D element. Rather, it applies a force to the element that produces the specified extension if that element is free to expand without generating internal forces. The software adds the computed force to the other forces in your model. Because most elements in a model are not free to expand, the extension value that you specify in the **1D Axial Deformation** dialog box may not be achieved during the analysis.

The **1D Axial Deformation** command corresponds to the Nastran DEFORM bulk data entry.

This graphic shows an example of an axial deformation load. (A) shows the **1D Axial Deformation** load that was defined on each of the 1D bar elements.



(B) shows the nodal displacement results. Notice the deformation that occurs at the bar element locations.



For more information, see Enforced axial deformations for 1D elements (Nastran).

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® 1D Axial Deformation** 
Simulation Navigator	Under the active solution, right-click **Loads® New Load® 1D Axial Deformation**

Nodal pressure loads

What is it?

In the Nastran and ANSYS environments, you can use the new **Nodal Pressure** command to create pressure loads where the magnitude of the pressure varies at each node. This new command complements the existing **Pressure** command which you can use to define a single pressure value for each element face.

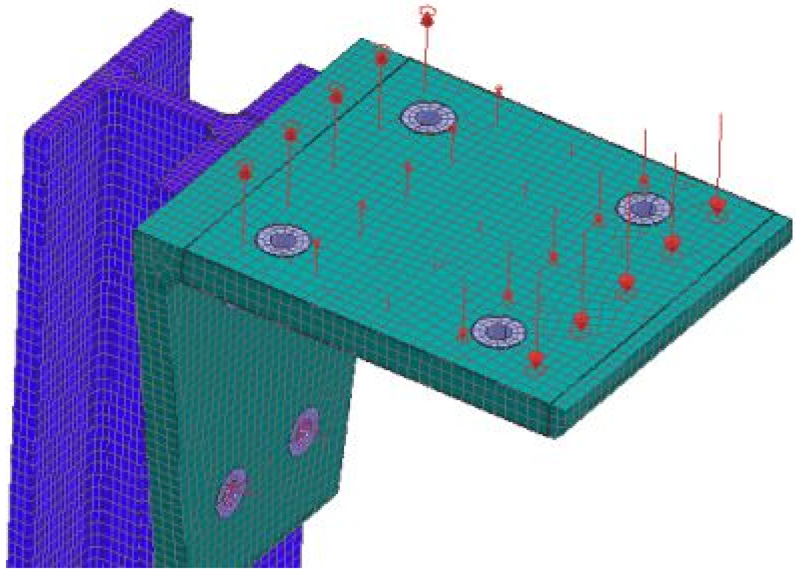
You can use the options in the **Nodal Pressure** dialog box to:

- Create a pressure load that is either normal to the surface of the element or a load that contains a traction component and is not normal to the surface

- apply a nonuniform pressure. You can define a different pressure value at each of the corner nodes to create a spatially varying pressure load.

The **Nodal Pressure** command corresponds to the Nastran PLOAD4 bulk data entry and to the ANSYS SFE,,,PRES command.

This graphic shows an example of a spatially varying nodal pressure load in the Nastran environment. Here, the pressure load varies linearly according to the X coordinate of each corner node.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran, MSC Nastran, or ANSYS as the specified solver
Toolbar	**Advanced Simulation® Nodal Pressure** 
Simulation Navigator	Under the active solution, right-click **Simulation Objects® New Load® Nodal Pressure**

Improved support for temperature definition in Nastran analyses

What is it?

This release includes several enhancements that improve the support for temperature definition in Nastran analyses.

New Material Temperatures command

You can use the new **Material Temperatures** command to specify the temperature or temperatures at which the software evaluates temperature-dependent material properties (MATTi bulk data entries).

When you include a **Material Temperatures** simulation object in a solution, Nastran evaluates any material properties for the selected nodes at the temperature you specify in the **Material Temperatures** dialog box rather than at the material's specified reference temperature.

- You can include only one **Material Temperatures** simulation object in a solution.

- You cannot include a **Material Temperatures** and an **Initial Temperatures** simulation object in the same solution.

The **Material Temperatures** command corresponds to the Nastran TEMPERATURE(MATERIAL) case control command.

In previous releases, there was no direct way to define a material temperature in NX.

New options for setting default temperatures in a solution

This release includes new options in the **Solution** and **Solution Step** dialog boxes that let you specify default initial, material, and load temperatures for all other nodes in your model for which you have not defined a temperature load. These options correspond to the TEMPD bulk data entry. You can use these new options to apply different temperature loads in each subcase.

Support for time-dependent default temperature loads in advanced nonlinear analyses

In NX Nastran advanced nonlinear solution steps, you can now define default temperature loads as being time-dependent. In the **Solution Step** dialog box, when you define the **Temperature Load Default (TEMPD)**, you can select the new **Field** option to use field to specify how the default temperature varies with time.

Where do I find it?

Material Temperature command

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® Material Temperatures**
Simulation Navigator	Under the active solution, right-click **Simulation Objects® New Simulation Object® Material Temperatures**

Temperature Load Default solution option

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Location in dialog box	In the **Solution** dialog box, click the **General** tab and enter a value in the **Initial Temperatures Default** or **Material Temperatures Default** boxes.

Temperature Load Default solution step option

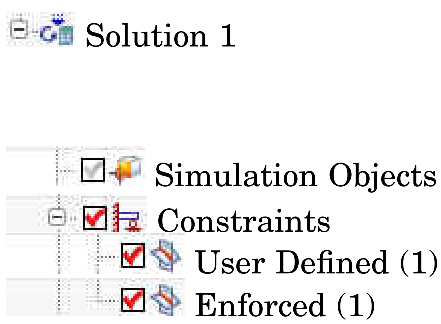
Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Location in dialog box	In the **Solution Step** dialog box, click the **General** tab and enter a value in the **Temperature Load Default** box.

New process and interface for resolving constraint conflicts

What is it?

Conflicting constraints occur when redundant constraints are defined for a given DOF. The process and user interface for resolving conflicting constraints has been simplified in NX 8 and the user interface has been redesigned.

Beginning in this release, the software only checks for constraint conflicts at the DOF level rather than at the nodal level. This ensures that only actual conflicts are reported. When a constraint conflict occurs, an icon now appears at the solution level in the **Simulation Navigator**:



To resolve the conflict, you can right-click the solution and choose the new **Resolve Constraints** command. You can then use the options in the **Constraint Resolution Manager** dialog box to resolve the conflict. The available options depend on the nature of the conflict. For example, you can:

- Specify which of the conflicting constraints you want to use.
- Create a constraint with new values at the point of conflict.
- Average or add the values between the conflicting constraints.
- Keep the overlapping constraints.

After you resolve a constraint, the software removes it from the **Conflicting Constraints** list and updates the associated constraints.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file and existing constraints
Simulation Navigator	Right-click the active solution and select **Resolve Constraints**

Adding boundary conditions to the active solution

What is it?

In this release, you can use the new **Add to active solution or step** command to quickly add loads, constraints, or simulation objects to the active solution or solution step. In previous releases, you had to drag individual loads and constraints into the appropriate solution or step **Simulation Navigator**, which could be cumbersome in larger models.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file, an existing solution, and multiple existing constraints, loads, or simulation objects
Simulation Navigator	Right-click one or more constraints and select **Add to active solution or step**

ANSYS thermal boundary condition enhancements

What is it?

This release includes several enhancements to the boundary conditions in the ANSYS thermal solver environment.

Heat Flux enhancements

The **Type** list in the **Heat Flux** dialog box has the following changes:

- The **Elemental** option now allows you to apply a heat flux to selected elements.

- The new **Element Face** option, which was erroneously labeled as **Elemental** in previous releases, allows you to apply a heat flux directly to the faces of selected elements.

Applying certain thermal boundary conditions to ANSYS surface effect elements

Beginning in this release, you can now have the software apply certain thermal boundary conditions to ANSYS surface effect elements (SURF151 and SURF 152) rather than to the regular thermal elements in your mesh. For example, you can use surface effect elements to model radiation between the surface and a point (an optional node for the SURF151 or SURF152 element).

Now, when you define a **Heat Flux**, **Heat Generation**, or **Convection** boundary condition, you can use the new **Add Surface Effect Elements** option to apply the loads to SURF151 or SURF152 elements instead of the existing thermal elements. When you export or solve your model, the software automatically creates the SURF151 and SURF152 elements.

See **Unsatisfied xref title** for more details on SURF151 and SURF152 element support.

Connecing certain thermal boundary conditions to a coupled thermal-fluid pipe element

If you select the new **Add Surface Effect Elements** option in the **Heat Flux**, **Heat Generation**, or **Convection** dialog box, you can also select the new **Connect to FLUID116 elements** option to attach the SURF151 or SURF152 elements to FLUID116 elements.

A FLUID116 element is an ANSYS 3D element with the ability to conduct heat and transmit fluid between its two primary nodes. You can use FLUID116 elements in steady-state or transient thermal analyses. For example, you can use surface effect elements to generate film coefficients and bulk temperatures from FLUID116 elements.

See **Unsatisfied xref title** for more details on FLUID116 element support.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver and **Thermal** as the specified analysis type
Toolbar	**Advanced Simulation® Heat Flux** or **Heat Generation** or **Convection**

External superelement system modeling

What is it?

Beginning with this release, you can replace individual components in your assembly FEM with external superelements to define a system model.To do this, you modify the component attributes to reference the binary results file (*_0.op2) from an NX Nastran SOL101 – Superelement or SOL103 – Superelement analysis.

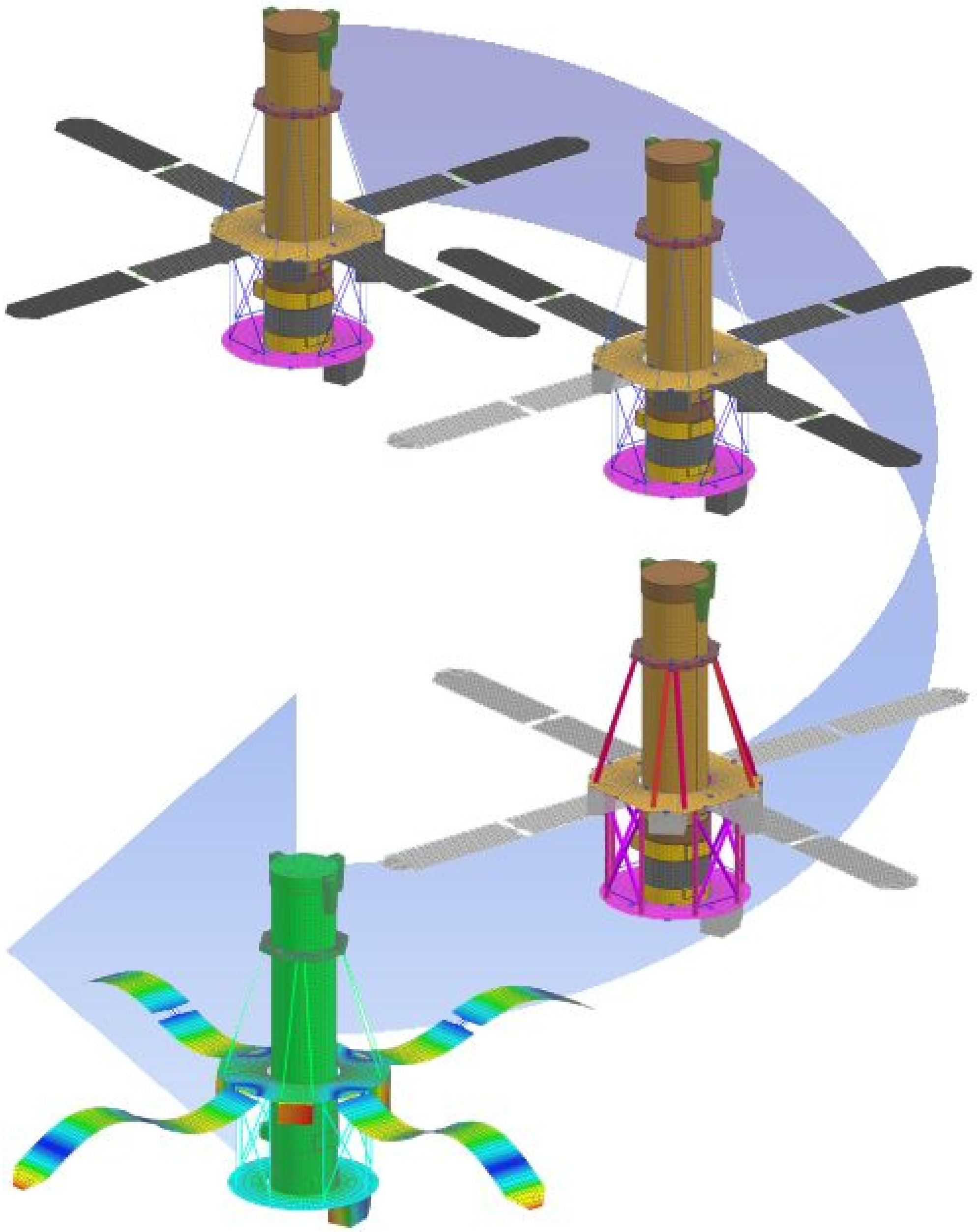
NX visually represents a superelement in your assembly FEM using either a lightweight, transparent, non-selectable approximation of the base FEM, or a generic superelement symbol showing the attachments nodes and center of mass. Any remaining components or connection elements in the assembly FEM constitute the residual of the system model.

Solving a system model

When you solve a system model, NX automatically recognizes that it references external superelements, and submits the Nastran input file with the appropriate statements to perform a system run.

Post-processing a system model

When you solve the system model, results for the residual are listed for the solution in the **Post-Processing Navigator**. Additional results for each superelement are shown as children of the residual results. When plotting residual results, choose the **Plot and Overlay Loaded SE** command to automatically overlay loaded superelement results, with consistent deformation and color bar scaling, in the same viewport.



Clockwise from top: An assembly FEM; a solar panel component replaced by a superelement; the completed system model – all solar panels and payload modules are represented by superelements; results from the solved system model.

Why should I use it?

Using external superelements, you can reduce the number of DOFs, and consequently the number of equations, to a more manageable number in an arbitrarily large model.

Where do I find it?

Representing a component FEM with a superelement:

Application	Advanced Simulation
Prerequisite	NX Nastran 8. An assembly FEM of the system being modeled. Static or modal reductions (*_0.op2 files) of components to be modeled as superelements.
Simulation Navigator	Component FEM or subassembly FEM node ® right-click ® **Edit Attributes**.
Location in dialog box	**FE Model Occurrence Attributes** dialog box ® **Representation** ® **Super Element**

Import, export, and solve enhancements

New customer default to control units for import and export

What is it?

This release includes new **Exporter** and **Importer** customer defaults. Use these new customer defaults se to control the default unit system the software uses to import and export solver input files for all supported solvers. Additionally, in the NX Nastran and MSC Nastran environments, the **Exporter** customer default controls the default unit system in the **Advanced Solver Options** dialog box.

Where do I find it?

Application	Advanced Simulation
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Simulation® General® Environment** tab

Controlling whether a solve aborts when the Model Setup Check detects errors

What is it?

You can now control whether a solve continues or aborts when the **Model Setup Check** command detects errors. In some cases, models can contain errors, such as incompletely defined data, and still be valid to solve. Now, you can use the new **Abort Solve on Model Setup Check Errors** customer default to control how the software handles detected errors.

- If you select this customer default, the software does not proceed with a solve when errors are present in the input file.
- If you clear the check box, the software tries to proceed with the solve even when errors are present in the input file.

This customer default applies to the Nastran, Abaqus, ANSYS, and LS-DYNA solver environments.

Where do I find it?

Application	Advanced Simulation
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Simulation® General® Environment** tab

Nastran support enhancements

Additional solve options available

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 Nastran input file. For example, you can:

- Specify the unit system in which to solve the model.
- Control the specific data that NX includes in the Nastran input file. You can, for example, choose to exclude entire sections, such as the entire file management section, from the input file. Within the bulk data section, you can use the **By card name** option in the **Filter Method** list to exclude selected bulk data entries from the solver input file.
- Specify offset values for the different IDs in the input file, such a grids or physical properties.

- Use the **Model Orientation** option to export the model in a different coordinate system than the one used in the Simulation file. This allows you to re-orient the model when you solve to a specified coordinate system.

Note If you modify any of the settings in the **Output Options** or **Bulk Data Section** group, the software does not perform a **Model Setup Check** before the solve.

In previous releases, these options were only available from the **Export Simulation** dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Menu	**Analysis® Solve**
Simulation Navigator	Right-click a solution and choose **Solve**

Support for Nastran transient heat transfer analyses

You can now perform Nastran transient heat transfer analysis in Advanced Simulation. When you create a thermal solution, you can use the new **NLTCSH 159** option in the **Solution** dialog box to create a SOL 159 solution. A SOL 159 analysis is similar to a SOL 153 analysis but allows you to analyze the temperatures and heat transfer as a function of time.

For more information, see the *NX Nastran Thermal Analysis User's Guide*.

New output requests

This release also includes new output options in the **Thermal Output Requests** dialog box to support SOL 159 analyses.

- You can use the options on the new **Enthalpy** tab to request enthalpy output. The options correspond to the describers for the ENTHALPY case control command.

- You can use the options on the new **Rate of Enthalpy Change** tab to request the rate of change of enthalpy output. The options correspond to the describers for the HDOT case control command.

Where do I find it?

Solution 159

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Simulation Navigator	Right-click the Simulation® **New Solution**
Menu	**Insert**® **Solution**

Additional thermal output requests

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation**® **Modeling Objects**  ® **Type** list® **Thermal Output Requests**
Menu	**Insert**® **Modeling Objects**® **Type** list® **Thermal Output Requests**

New checks for unintentional grounding and mass reduction

You can use the new **Rigid Body Checks** modeling object to check the motion grounding and mass reduction of rigid bodies in your solution.

Note If you include a **Rigid Body Checks** modeling object in your solution, you must use the Nastran sparse solver.

In the **Rigid Body Checks** dialog box, you can:

- Use the **Grounding Check** options to evaluate selected DOF sets and identify unintentional constraints and ill-conditioning in the stiffness matrix. The options on the **Grounding Check** tab correspond to the describers for the Nastran GROUNDCHECK case control command.

- Use the **Mass Reduction Check** options to compute the rigid body mass at each stage of the mass matrix reduction and compare it with the rigid body mass of the g-set. The options on the **Mass Reduction Check** tab correspond to the describers for the Nastran WEIGHTCHECK case control command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation**® **Modeling Objects**  ® **Type** list® **Rigid Body Checks**
Menu	**Insert**® **Modeling Objects**® **Type** list® **Rigid Body Checks**

Improved support for printing bulk data in output

Use the new **Bulk Data Echo Request** modeling object to control whether Nastran prints the bulk data section of your input file in the results file. For example, you can control:

- Whether the software prints the bulk data to the .f06 results file, to the ASCII punch (.pch) file, or to both files.

- Whether the software prints updated bulk data from optimization (SOL 200) analyses.

For example, if you use a **Bulk Data Echo Request** to print out the updated bulk data output from a SOL 200 analysis, you can then merge that bulk data into the input file for a subsequent analysis.

The options in the **Bulk Data Echo Request** dialog box correspond to the describers for the Nastran ECHO case control command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Nastran solution
Toolbar	**Advanced Simulation**® **Modeling Objects**  ® **Type** list® **Bulk Data Echo Request**
Menu	**Insert**® **Modeling Objects**® **Type** list® **Bulk Data Echo Request**

Support for glue results

What is it?

This release includes improved support for results of analyses that include glue boundary conditions. In this release, you can now:

- Request the output of NX Nastran glue results directly from NX.
- Post-process NX Nastran glue results.

Requesting glue results output

Use the options on the new **Glue Result** tab in the **Structural Output Requests** dialog box to recover glue surface forces and tractions for solid and shell elements. When you create a **Glue Result** output request, the software creates a BGRESULTS case control command in your NX Nastran input file.

- For **Surface-to-Surface Gluing** conditions, you can recover forces or tractions.
- For **Edge-to-Surface Gluing** conditions, you can only recover forces, not tractions.

You can use a **Glue Result** output request to obtain glue results in SOL 101, 103, and 105 analyses.

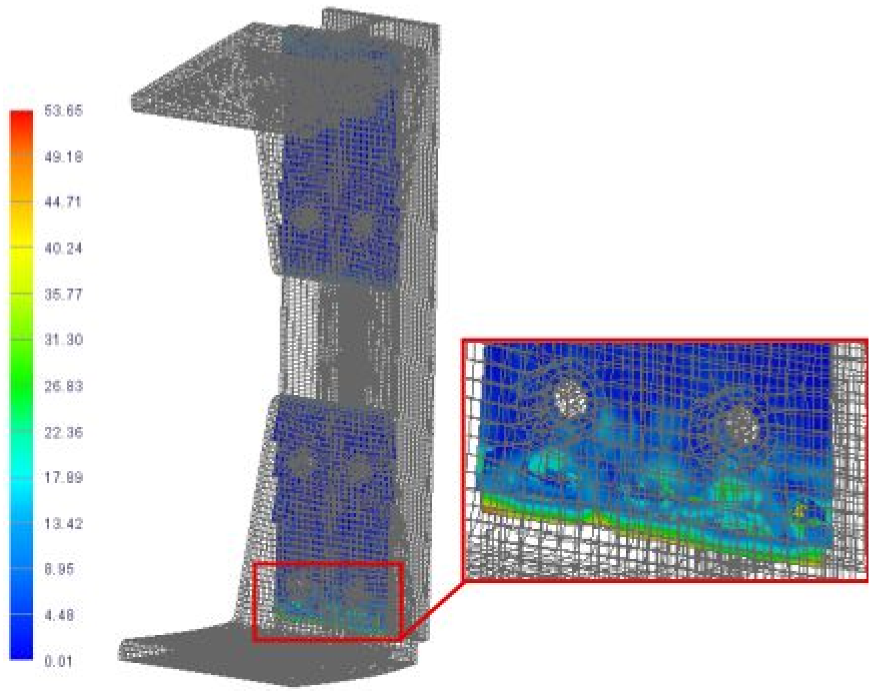
The BGRESULTS command is available in the NX Nastran 8.0 release and is similar to the BCRESULTS command for recovering contact forces. With the BGRESULTS command, NX Nastran calculates and stores glue tractions at the nodes that are located on the glued surfaces. The new glue tractions, which are similar to the existing contact results, are calculated and stored at the nodes that are located on the glue surfaces. The normal component of the tractions is a scalar value while the in-plane (tangential) tractions are output in the basic coordinate system.

Note For surfaces on which you have created an **Edge-to-Surface Gluing** definition, NX Nastran recovers only point forces and not surface tractions.

You can use the new **Glue Result** customer default to control whether the **Enable BGRESULTS Request** option on the **Glue Result** tab is selected by default.

Post-processing support for glue results

This release also includes post-processing support for glue forces and surface tractions. This graphic shows an example of a contour plot display of glue traction results.



Where do I find it?

Glue Result output request

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran as the specified solver
Toolbar	**Advanced Simulation**® **Modeling Objects** ® **Type** list® **Structural Output Requests**
Menu	**Insert**® **Modeling Objects** ® **Type** list® **Structural Output Requests**

Glue Result customer default

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran as the specified solver
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Simulation**® **NASTRAN**® **Solution** tab

Enhancements for specifying units on import

What is it?

In the Nastran environment, the **Import Solution** dialog box contains enhancements to how you specify the units used in the .dat or .op2 file you are importing.

Improved presentation for the input file's structural units

In the **Import Simulation** dialog box, use the **Default Units** list to specify the unit system used in the input file that you are importing. When NX imports the file, the software automatically converts these units to the appropriate CAE base units.

In previous releases, the different **Default Units** options listed the force and length units for the unit system. In this release, the options now also list the mass units for the unit system. This makes it easier for you to differentiate between the available choices.

Ability to import temperatures in a different unit system

This release also includes a new **Specify Temperature Units** check box.

- If you select the check box, you can then specify a different unit system for the temperatures in your model. This option is useful when the temperature values in your model are defined in a different unit system than the structural unit system.

- If you do not select the check box, the software imports the temperature values in the unit system you specified in the **Default Units** list.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Import® Simulation**

NX Nastran subcase manager

This release includes a new **Subcase Association Manager** for NX Nastran solutions. You can use the **Subcase Association Manager** to view and manage the loads used in each subcase. From the **Subcase Association Manager** dialog box, you can activate and deactivate individual loads. As you make changes to the loads, constraints, or simulation objects assigned to each subcase, the **Simulation Navigator** updates to reflect the changes.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran as the specified solver
Simulation Navigator	Right-click a selected solution® **Subcase Manager**

Improved round-trip capabilities for bulk data

The term *round-trip* refers to the process of data conversion that occurs when you import a solver data file into NX, and then export the same file out of NX. This release offers some initial improvements to how NX preserves the order and format of Nastran data through the round-trip process.

For this release, the round-trip capabilities are limited to:

- ASCII format files, such as .dat files.
- Selected entries within the bulk data section of the Nastran input file.

Round-trip process

When you import an ASCII format Nastran file, you can use the new **Create Round Trip Parameters modeling object** option in the **Import Simulation** dialog box to indicate that you want to preserve the format of your data for future export. If you select this option, the software then creates a **Round Trip Parameters** modeling object which stores round trip data in the associated FEM file.

The **Round Trip Parameters** modeling object is primarily used by NX to store data about the input file. You cannot edit the content of a **Round Trip Parameters** modeling object or create a new one. However, you can use the **Round Trip Parameters** dialog box to specify a limited number of options that control how the software uses the data. You can specify, for example:

- Whether you want to preserve the sequence of the bulk data entries.
- Whether you want to preserve the field format of the bulk data entries.

The software associates the **Round Trip Parameters** modeling object with the related solution. On the general **Tab** of the **Solution** dialog box, the **Round Trip Parameters** option lists the associated modeling object.

When you export the model, the **Export Simulation** dialog box lists the related **Round Trip Parameters** modeling object in the **Formatting Options** section.

Bulk data entries supported

Currently, not all bulk data entries can be preserved exactly through the data round trip process. In general, this release supports the round trip of the following types of bulk data entries:

- GRID entries
- Element entries
- Material entries
- Physical property entries
- BSURF, BSURFS, and BLSEG entries

For a complete list of bulk data entries that are supported for the round trip process, see Preserving the order and format of bulk data through the round-trip process.

Customer default

This release also includes a new customer default that you can use to set the default state of the **Create Round Trip Parameters modeling object** option in the **Import Simulation** dialog box.

Where do I find it?

Round Trip Parameters modeling object

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects**  ® **Type** list® **Round Trip Parameters**
Menu	**Insert® Modeling Objects® Type** list® **Round Trip Parameters**

Round Trip Parameters customer default

Application	Advanced Simulation
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Simulation® General® Environment** tab

Descriptions now exported as comments

When you enter text in the **Description** box in the loads, constraints, and simulation object dialog boxes, NX now exports those descriptions as comments in your Nastran input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with Nastran as the specified solver
Menu	**File® Export® Simulation** or **Analysis® Solve**

Changes to SOL 103 and 106 solutions

This release includes changes to the SOL 103 and SOL 106 solutions within NX.

SOL 103 changes

When you create an **SEMODES 103** type solution, NX now automatically creates a **Subcase-Eigenvalue Method** type of solution step. This solution step is intended to contain the **Real Eigenvalue-Lanczos** or **Real Eigenvalue-Householder** modeling object. You use those modeling objects to specify the real eigenvalue extraction parameters for the METHOD case control command.

By default, each **Subcase-Eigenvalue Method** has a **Real Eigenvalue-Lanczos** modeling object automatically assigned to it. However, you can modify the modeling object to specify different options or specify a **Real Eigenvalue-Householder** modeling object instead. Additionally, you can still specify an eigenvalue modeling object at the solution level if you want to specify a global eigenvalue method for multiple dynamic subcases.

Note This enhancement was first introduced in NX 7.5.2 but was not documented.

SOL 106 changes

In this release, the **NLSTATIC 106** solution type has been divided into two different solution types:

- **NLSTATIC 106–Single Constraint**, which lets you create subcases with unique loads but with identical constraints for each subcase in the solution.

- **NLSTATIC 106–Multi Constraint**, which lets you create subcases with unique loads and unique constraints.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Simulation Navigator	Right-click the Simulation® **New Solution**
Menu	**Insert**® **Solution**

Axial deformation loads for 1D elements

What is it?

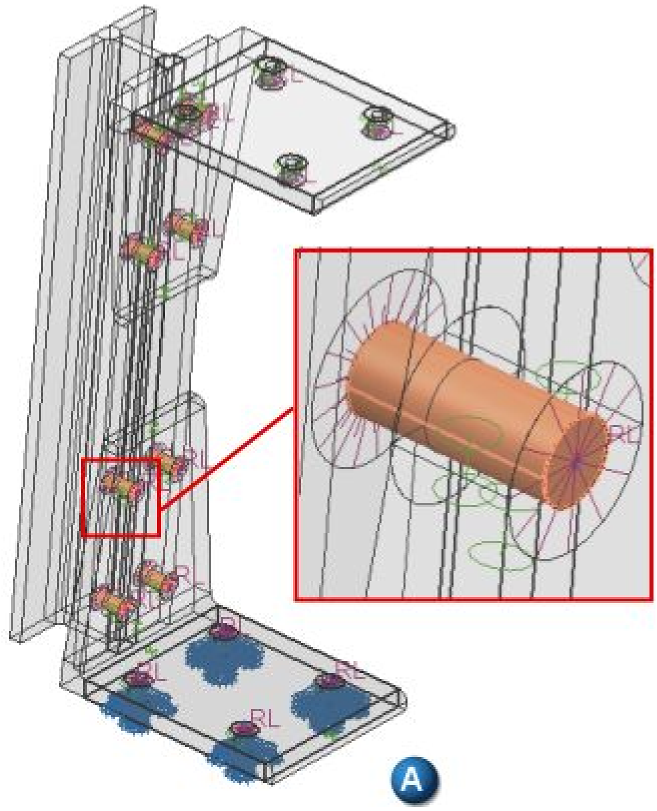
In the Nastran environment, you can now use the new **1D Axial Deformation** command to apply an enforced axial deformation (deflection) to CBAR, CBEAM, CONROD, CROD, and CTUBE type elements.

In NX, you can use the **1D Axial Deformation** command to apply axial deflections in Solution 101, 105, and 200 type analyses. Enforced deflections are useful, for example, when you analyze misfit or misalignment conditions in structures.

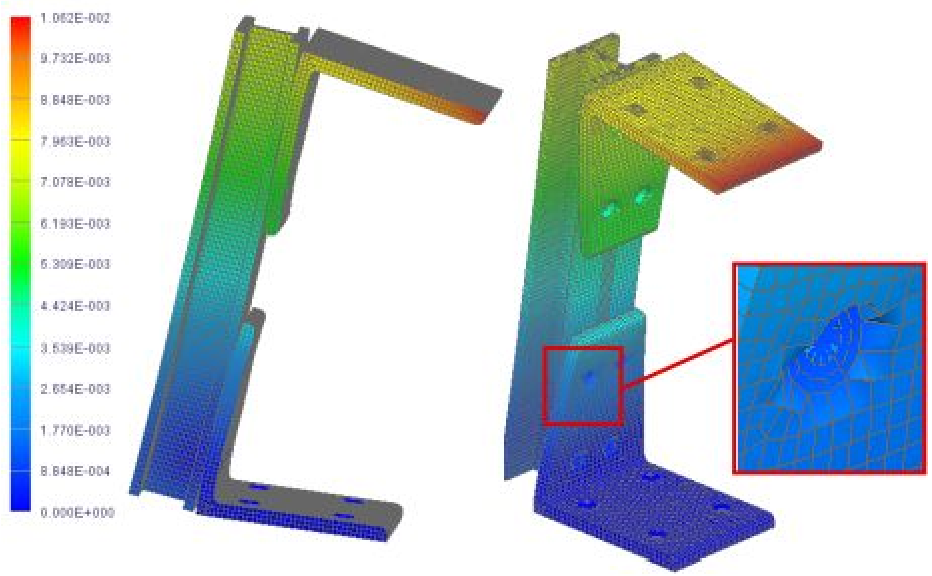
The **1D Axial Deformation** command does not enforce a strain or apply an extensional length to the 1D element. Rather, it applies a force to the element that produces the specified extension if that element is free to expand without generating internal forces. The software adds the computed force to the other forces in your model. Because most elements in a model are not free to expand, the extension value that you specify in the **1D Axial Deformation** dialog box may not be achieved during the analysis.

The **1D Axial Deformation** command corresponds to the Nastran DEFORM bulk data entry.

This graphic shows an example of an axial deformation load. (A) shows the **1D Axial Deformation** load that was defined on each of the 1D bar elements.



(B) shows the nodal displacement results. Notice the deformation that occurs at the bar element locations.



B

For more information, see Enforced axial deformations for 1D elements (Nastran).

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® 1D Axial Deformation** 
Simulation Navigator	Under the active solution, right-click **Loads® New Load® 1D Axial Deformation**

Nodal pressure loads

What is it?

In the Nastran and ANSYS environments, you can use the new **Nodal Pressure** command to create pressure loads where the magnitude of the pressure varies at each node. This new command complements the existing **Pressure** command which you can use to define a single pressure value for each element face.

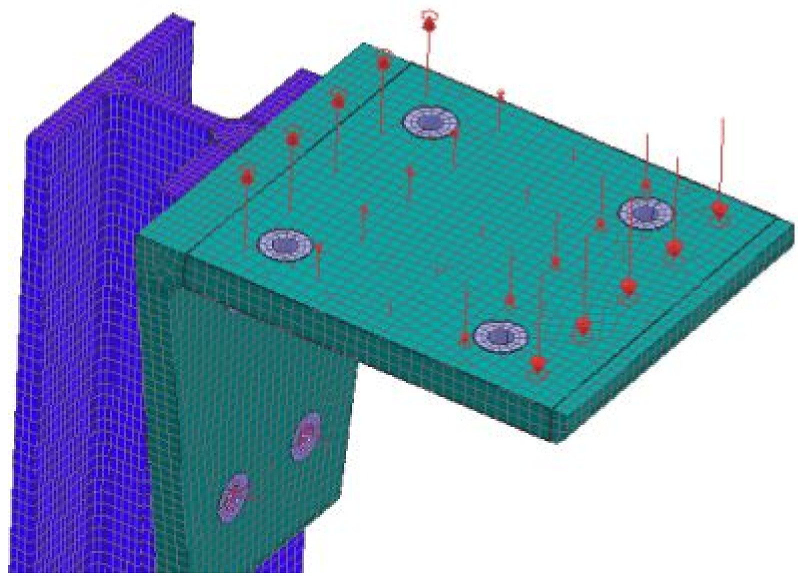
You can use the options in the **Nodal Pressure** dialog box to:

- Create a pressure load that is either normal to the surface of the element or a load that contains a traction component and is not normal to the surface

- apply a nonuniform pressure. You can define a different pressure value at each of the corner nodes to create a spatially varying pressure load.

The **Nodal Pressure** command corresponds to the Nastran PLOAD4 bulk data entry and to the ANSYS SFE,,,PRES command.

This graphic shows an example of a spatially varying nodal pressure load in the Nastran environment. Here, the pressure load varies linearly according to the X coordinate of each corner node.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran, MSC Nastran, or ANSYS as the specified solver
Toolbar	**Advanced Simulation® Nodal Pressure**
Simulation Navigator	Under the active solution, right-click **Simulation Objects® New Load® Nodal Pressure**

Improved support for temperature definition in Nastran analyses

What is it?

This release includes several enhancements that improve the support for temperature definition in Nastran analyses.

New Material Temperatures command

You can use the new **Material Temperatures** command to specify the temperature or temperatures at which the software evaluates temperature-dependent material properties (MATTi bulk data entries).

When you include a **Material Temperatures** simulation object in a solution, Nastran evaluates any material properties for the selected nodes at the temperature you specify in the **Material Temperatures** dialog box rather than at the material's specified reference temperature.

- You can include only one **Material Temperatures** simulation object in a solution.
- You cannot include a **Material Temperatures** and an **Initial Temperatures** simulation object in the same solution.

The **Material Temperatures** command corresponds to the Nastran TEMPERATURE(MATERIAL) case control command.

In previous releases, there was no direct way to define a material temperature in NX.

New options for setting default temperatures in a solution

This release includes new options in the **Solution** and **Solution Step** dialog boxes that let you specify default initial, material, and load temperatures for all other nodes in your model for which you have not defined a temperature load. These options correspond to the TEMPD bulk data entry. You can use these new options to apply different temperature loads in each subcase.

Support for time-dependent default temperature loads in advanced nonlinear analyses

In NX Nastran advanced nonlinear solution steps, you can now define default temperature loads as being time-dependent. In the **Solution Step** dialog box, when you define the **Temperature Load Default (TEMPD)**, you can select the new **Field** option to use field to specify how the default temperature varies with time.

Where do I find it?

Material Temperature command

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® Material Temperatures**
Simulation Navigator	Under the active solution, right-click **Simulation Objects® New Simulation Object® Material Temperatures**

Temperature Load Default solution option

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Location in dialog box	In the **Solution** dialog box, click the **General** tab and enter a value in the **Initial Temperatures Default** or **Material Temperatures Default** boxes.

Temperature Load Default solution step option

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Location in dialog box	In the **Solution Step** dialog box, click the **General** tab and enter a value in the **Temperature Load Default** box.

Import and export support enhancements

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
BGRESULTS case control command	Yes	No	See **Unsatisfied xref title** for more information.
DEFORM bulk data entry	Yes	Yes	See **Unsatisfied xref title** for more information.
ENTHALPY case control command	Yes	No	See **Unsatisfied xref title** for more information.
GROUNDCHECK case control command	Yes	No	See **Unsatisfied xref title** for more information.
HDOT case control command	Yes	No	See **Unsatisfied xref title** for more information.

MATCID bulk data entry	Yes	No	The BY field is not supported for import or export.
MATFT	Yes	No	When you import into NX, you can have a single MATFT definition per material. In Nastran, you can use the MATFT entry to define different allowable values for the various failure theories. In NX, only the first one is retained. Additionally, for import, the F13 and F23 fields are unsupported. If MATFT corresponds to MAT9 then entries F12, Xc, Yc, Zc, Xt, Yt, Zt, S12, S13 and S23 are also unsupported.
PCOMPS	Yes	No	When you import into NX, if SOUT=YES for any one ply, then SOUT=YES for all plies. TFLAG REL is not supported.
PLOAD4	Yes	Yes	See **Unsatisfied xref title** for more information.
WEIGHTCHECK case control command	Yes	Yes	See **Unsatisfied xref title** for more information.

Parameters

The following table details the changes in parameter support for this release.

Parameter	Description	Notes
CNTASET	When contact conditions are defined in a SOL 101 analysis, CNTASET determines whether a static condensation reduction (A-set) occurs with the contact degrees-of-freedom.	This parameter is now supported for import and export.
LMDYN, LMSTAT	With Lagrange RBE3 elements, these parameters scale the artificial stiffness term c_k.	This parameter is now supported for import and export.
OEPT	Disables the output of the EPT data block to the .op2 file.	This parameter is now supported for import and export.
OMPT	Disables the output of the MPT data block to the .op2 file.	This parameter is now supported for import and export.

RMXPANEL	When the RMAXMIN case control command is defined in a SOL 112 solution, this parameter can be used to process the data recovery in a series of smaller "panels" rather than the entire output set at once. This will reduce the overall amount of scratch disk space required for the run.	This parameter is now supported for import and export.
SDAMPUP	In a modal frequency response solution (SOL 111), this parameter determines whether modal damping is updated for the response calculation at each frequency.	This parameter is now supported for import and export.
XFLAG	Controls the equation that Nastran uses to calculate the element strain energy for linear elements.	The XFLAG parameter now defaults to a value of 2.

System cell support updates

The following table details the changes in system cell support for this release.

System Cell	Name	Description	Notes
422	ENFMOTN	Controls which formulation is used for enforced motion response analysis (and mode acceleration, if requested).	The default value for this cell is now 0: the constraint mode method of enforced motion formulation (and new mode acceleration method).
492		Determines whether the factor caching for the Lanczos real eigensolver is used.	This system cell is now supported for import and export.

503		On the ACSRCE bulk entry, determines if the scale factor A is included inside or outside of the radical. See the ACSRCE bulk entry for the source strength equation.	This system cell is now supported for import into NX.
516		When gluing solid element faces, determines if the solid element face normal check occurs. See the remarks for the BGSET case control command for more information.	This system cell is now supported for import into NX.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Import® Simulation** **File® Export® Simulation**

Abaqus support enhancements

Keyword support enhancements

What is it?

This release includes support for a number of new Abaqus keywords as well as enhancements to previously supported keywords.

Keyword	Supported parameters	Import support	Export support	Notes
`*CHANGE FRICTION`	All parameters are supported for import and export.	Yes	Yes	In the NX 7.5.2 release, `*CHANGE FRICTION` was only supported for export.
`*CREEP`	All parameters are supported for import and export.	Yes	Yes	In previous releases, *CREEP was only supported for export.
`*ELCOPY`	All parameters supported.	Yes	No	

Keyword	Supported parameters	Import support	Export support	Notes
`*FILM`	The `FPOS` and `FNEG` parameters are now supported	Yes	Yes	For more information, see **Unsatisfied xref title**.
`*FREQUENCY`	All parameters except `AMS` are supported.	Yes	Yes	The `NORMALIZATION` parameter is now supported for import.
`*SHEAR TEST DATA`	All parameters are supported for import and export.	Yes	No	Support for `*SHEAR TEST DATA` has been added in conjunction with support for `*VISCOELASTIC` so that you can use shear test data to define viscoelastic materials.
`*SHELL SECTION`	The parameters `POISSON`, `STACK DIRECTION=ORIENTATION`, and `THICKNESS MODULUS` are now supported for import and export.	Yes	Yes	For more information on the `STACK DIRECTION=ORIENTATION` enhancement, see **Unsatisfied xref title**.
`*SURFACE BEHAVIOR`	The `PRESURE-OVERCLOSURE=LINEAR` parameter is now supported for export.	Yes	Yes	
`*VISCOELASTIC`	All parameters are supported for import and export.	Yes	Yes	In previous releases, `*VISCOELASTIC` was only supported for export.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with Abaqus as the specified solver
Menu	**File→Import→Simulation**

Enhancements for Abaqus element-based film conditions

What is it?

In the Abaqus environment, you can use the new **Free Convection on 2D Element Faces** option in the **Convection** dialog box to define sink temperatures and film convection coefficients for a heat transfer analysis. In Abaqus, film conditions define heating or cooling due to convection by surrounding fluids.

With the new **Free Convection on 2D Element Faces** option, you define a sink temperature value, Θ^0, and a film coefficient, h, on the faces of 2D elements. You can also use the new **Film Type** option to control the direction in which the software applies the film condition.

- The **FPOS** option applies the film condition to the top face of the element in the positive normal direction.

- The **FNEG** option applies the film condition to the bottom face of the element in the negative normal direction.

In previous releases, although you could use the **Convection** command to apply a film condition to the faces of 2D elements, you could not control the direction in which the condition was applied. Previously, the software always applied the condition in the elements' positive normal direction.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with Abaqus as the specified solver and **Thermal** as the specified **Analysis Type**
Toolbar	**Advanced Simulation® Convection**
Simulation Navigator	Right-click **Constraints** ® **New Constraint** ® **Convection**

Laminates stacking direction now based on a local orientation

What is it?

Beginning in this release, NX now supports the `STACK DIRECTION=ORIENTATION` parameter for the Abaqus `*SHELL SECTION` keyword. This means that in a laminates model in the Abaqus environment, you can now specify a user-defined orientation for a laminate stacking sequence. In previous releases, NX only supported a stacking direction that was based on an element's isoparametric directions.

To specify a local orientation for a laminate from the **Basic Laminate** physical property dialog box:

1. On the **General** tab, in the **Continuum Shell** list, select **Yes**.

2. In the **Stack Direction Method** list, select **Local Orientation**.

3. Select the **Set Material Orientation** option and specify the material orientation.

To specify a local orientation for a laminate from the **Laminate Modeler** dialog box:

1. In the **Properties** group, in the **Continuum Shell** list, select **Yes**.

2. In the **Stack Direction Method** list, select **Local Orientation**.

3. Select the **Set Material Orientation** option and specify the material orientation.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with Abaqus as the specified solver
Toolbar	**Laminates** ® **Laminate Physical Property**
Menu	**Insert** ® **Laminate** ® **Physical Property**

Improvements for isotropic plastic material support

What is it?

This release includes extended support of temperature-dependent stress-strain data for isotropic plastic materials in the Abaqus environment.

In previous releases, NX correctly wrote out temperature-dependent plastic materials with isotropic hardening to the Abaqus input file only when the hardening curve was defined with stress vs. plastic strain data points.

In this release, if the material's definition includes stress vs. strain data at various temperatures and a yield strength vs. temperature curve, NX processes both sets of data and writes out the temperature-dependent stress vs. plastic strain data to the Abaqus input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with Abaqus as the specified **Solver Environment**
Toolbar	**Advanced Simulation® Manage Materials** 

Import support improvements for output variables

What is it?

This release includes improved import support for the Abaqus output variables that you can select from the **Abaqus Structural Output Requests** and **Abaqus Structural Output Requests** dialog boxes. Now, when you import an Abaqus input file, the software imports:

- All Abaqus output variables that are supported in NX. In previous releases, all the output variables were supported for export, but only a small subset were supported for import.

- Any group references (defined with the Abaqus `*ELSET` and `*NSET` keywords) that are associated with those output variables.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with Abaqus as the specified solver
Menu	**File→Import→Simulation**

ANSYS support enhancements

Support for additional thermal element types

What is it?

NX now supports additional types of ANSYS thermal elements.

Support for FLUID116 elements

In the ANSYS thermal environment, you can now create FLUID116 elements from the **1D Mesh** dialog box. The FLUID116 element is a coupled thermal-fluid pipe element. It can conduct heat and transmit fluid between its primary (required) nodes.

The FLUID166 element has two required nodes and two additional optional nodes that are necessary to define convection. In NX, when you create a mesh of FLUID116 elements from the **1D Mesh** dialog box, the software automatically creates the two required nodes for each element. You can then define the optional nodes (K and L) in the **Mesh Associated Data** dialog box.

In ANSYS, you can define additional properties for FLUID116 elements with real constants and KEYOPTS. In NX, you define these additional properties with:

- The new **FLUID116 ET** type of modeling object, which lets you set the KEYOPTS.

  NX currently does not support specification of the following coefficients as material properties.

  - The film coefficient (`MP, HF` or `TB, HFLM`).
  - The fluid conductance coefficient (`TB, FCON`)
  - The coefficient of friction (`MP, MU`).

  Therefore, in the **FLUID116 ET** dialog box, in the **KEYOPT(4)**, **KEYOPT(6)**, and **KEYOPT(7)** lists, if you select an option that depends on those material property values, you must manually add those properties to your ANSYS input file with a **User Defined Text** modeling object.

- The new **FLUID116** physical property table, which lets you set the real constants for the element.

Support for SURF151 and SURF152 elements

NX now supports both SURF151 and SURF152 type elements. SURF151 and SURF152 elements are thermal surface effect elements.

In ANSYS, surface effect elements are overlaid like a skin on top of the faces of other 2D or 3D thermal elements. You can use surface effect elements, for example, to generate film coefficients and bulk temperatures from FLUID116 elements and to model radiation to a point.

SURF151 and SURF152 elements also have an optional node that you can use to connect those elements with a FLUID116 element. This allows you to base the bulk temperature in convection calculations on a simplified representation of the fluid flow.

You do not directly create SURF151 and SURF152 elements in NX. When you define a **Heat Flux**, **Heat Generation**, or **Convection** boundary condition, you can specify whether you want the software to apply the loads to SURF151 or SURF152 elements instead of the existing thermal elements. When you export or solve your model, the software automatically creates the SURF151 and SURF152 elements.

In ANSYS, you can define additional properties for SURF151 and SURF152 elements with real constants and KEYOPTS. In NX, you define these additional properties with:

- The new **SURF151 ET** and **SURF152 ET** modeling objects, which let you set the KEYOPTS for the elements.

- The new **SURF151 Real Constants** and **SURF152 Real Constants** types of modeling object, which let you set the real constants.

Note If you import and ANSYS input file into NX that contains SURF151 or SURF152 elements, NX does not import them into the FEM file as thermal element. Rather, NX imports them into the Simulation file and associates the SURF151 or SURF152 elements with the appropriate **Heat Flux**, **Heat Generation**, or **Convection** boundary condition.

Support for PLANE55 and PLANE77 elements in the thermal environment

PLANE55 and PLANE77 elements are now supported in the ANSYS thermal environment. In previous releases, they were supported only in the ANSYS axisymmetric thermal environment. You can use the **Add Surface Effect Elements** option in the **Heat Flux**, **Heat Generation**, or **Convection** dialog boxes to overlay PLANE55 and PLANE77 elements with SURF151 or SURF152 elements.

Note When you create PLANE55 and PLANE77 elements in the thermal environment rather than the axisymmetric thermal environment, you must still create the elements on the XY plane. If you do not create the PLANE55 and PLANE77 elements, you will not be able to solve the model in ANSYS.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with ANSYS as the specified solver and **Thermal** as the specified analysis type

ANSYS thermal boundary condition enhancements

What is it?

This release includes several enhancements to the boundary conditions in the ANSYS thermal solver environment.

Heat Flux enhancements

The **Type** list in the **Heat Flux** dialog box has the following changes:

- The **Elemental** option now allows you to apply a heat flux to selected elements.

- The new **Element Face** option, which was erroneously labeled as **Elemental** in previous releases, allows you to apply a heat flux directly to the faces of selected elements.

Applying certain thermal boundary conditions to ANSYS surface effect elements

Beginning in this release, you can now have the software apply certain thermal boundary conditions to ANSYS surface effect elements (SURF151 and SURF 152) rather than to the regular thermal elements in your mesh. For example, you can use surface effect elements to model radiation between the surface and a point (an optional node for the SURF151 or SURF152 element).

Now, when you define a **Heat Flux**, **Heat Generation**, or **Convection** boundary condition, you can use the new **Add Surface Effect Elements** option to apply the loads to SURF151 or SURF152 elements instead of the existing thermal elements. When you export or solve your model, the software automatically creates the SURF151 and SURF152 elements.

See **Unsatisfied xref title** for more details on SURF151 and SURF152 element support.

Connecing certain thermal boundary conditions to a coupled thermal-fluid pipe element

If you select the new **Add Surface Effect Elements** option in the **Heat Flux**, **Heat Generation**, or **Convection** dialog box, you can also select the new **Connect to FLUID116 elements** option to attach the SURF151 or SURF152 elements to FLUID116 elements.

A FLUID116 element is an ANSYS 3D element with the ability to conduct heat and transmit fluid between its two primary nodes. You can use FLUID116 elements in steady-state or transient thermal analyses. For example, you can use surface effect elements to generate film coefficients and bulk temperatures from FLUID116 elements.

See **Unsatisfied xref title** for more details on FLUID116 element support.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver and **Thermal** as the specified analysis type
Toolbar	**Advanced Simulation® Heat Flux** or **Heat Generation** or **Convection**

Nodal pressure loads

What is it?

In the Nastran and ANSYS environments, you can use the new **Nodal Pressure** command to create pressure loads where the magnitude of the pressure varies at each node. This new command complements the existing **Pressure** command which you can use to define a single pressure value for each element face.

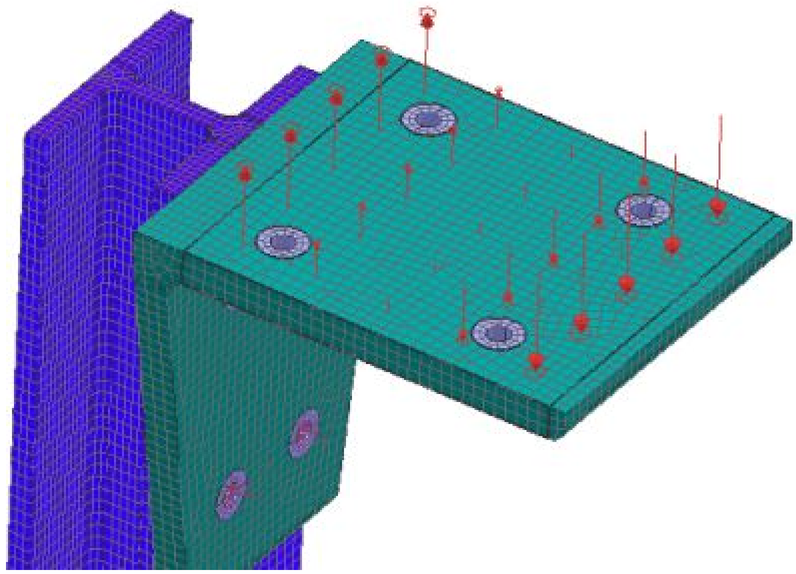
You can use the options in the **Nodal Pressure** dialog box to:

- Create a pressure load that is either normal to the surface of the element or a load that contains a traction component and is not normal to the surface

- apply a nonuniform pressure. You can define a different pressure value at each of the corner nodes to create a spatially varying pressure load.

The **Nodal Pressure** command corresponds to the Nastran PLOAD4 bulk data entry and to the ANSYS SFE,,,PRES command.

This graphic shows an example of a spatially varying nodal pressure load in the Nastran environment. Here, the pressure load varies linearly according to the X coordinate of each corner node.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran, MSC Nastran, or ANSYS as the specified solver
Toolbar	**Advanced Simulation® Nodal Pressure** 
Simulation Navigator	Under the active solution, right-click **Simulation Objects® New Load® Nodal Pressure**

LS-DYNA support enhancements

Including comments or files in an LS-DYNA input file

What is it?

When you export an LS-DYNA input file from NX, you can now insert specific comments, commands, or include files. Use the new **User Defined Text** modeling object to add either text or an entire file to a particular solution or step. If you choose to include a file, the software uses the LS-DYNA `*INCLUDE` keyword to insert the file:

- At the beginning of the input file, after the `*KEYWORD` keyword.
- At the end of the input file, before the `*END` keyword.

For example, you can use the **User Defined Text** modeling object to insert an external file that contains a portion of your LS-DYNA input file. The file can contain, for example, model definition data, comment lines, or other references to external files. You can also use a **User Defined Text** modeling object to add LS-DYNA keywords to your input file that are not currently supported in NX.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with LS-DYNA as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects**
Menu	**Insert® Modeling Objects**

Support for solution options and output requests

What is it?

This release includes enhancements that allow you to define the first input block of an LS-DYNA input file from NX. In an LS-DYNA input file, you use the first input block to define solution control options and output parameters. This complements the LS-DYNA enhancements that were included in the NX 7.5.2 release, which allowed you to define the second input block of an LS-DYNA input file (the model geometry, mesh, and material data).

Defining solution control options

NX now supports the following LS-DYNA keyword control cards:

- `*CONTROL_TERMINATION`, which you can use to specify options that control the termination time of a time step.
- `*CONTROL_ENERGY`, which you can use to control the computation of energy dissipation.
- `*CONTROL_HOURGLASS`, which you can use to specify an hourglass formulation that is different from the defaults.

You can use the new **Controls** modeling object to specify parameters for the keyword control cards. You can then use the new ***CONTROL** option in the **Solution** dialog box to include that modeling object in a specific solution.

Specifying output parameters

This release also includes support for several `*DATABASE` keyword cards. You use the `*DATABASE` keyword to create the output files that contain the LS-DYNA results information. NX now supports the following `*DATABASE` keyword cards:

- `*DATABASE_HISTORY`, which controls the nodes and elements that the software outputs to the `D3THDT` binary history file, as well as to the various ASCII files, such as `NODOUT`, `ELOUT`, `SPHOUT`.

- *DATABASE_BINARY, which controls the creation of binary output data. In NX, you can currently create D3PLOT and D3THDT binary files.
- *DATABASE_GLSTAT, which contains global statistics.
- *DATABASE_NODOUT, which contains nodal point data.
- *DATABASE_RWFORC, which contains wall forces.

After you create a **Database** modeling object, you can then use the new **Output Requests (*DATABASE)** option in the **Solution** dialog box to include that modeling object in a specific solution.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with LS-DYNA as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects**
Menu	**Insert® Modeling Objects**

Post-processing

General post-processing enhancements

What is it?

This release includes several enhancements to post-processing to improve the appearance of post views and to facilitate working with multiple post views.

Scalable fonts for headers, color bars, and markers

Post-processing displays now use standard NX scalable fonts for the post view header and color bar, and for marker displays. These fonts scale for optimal readability when you resize the graphics window.

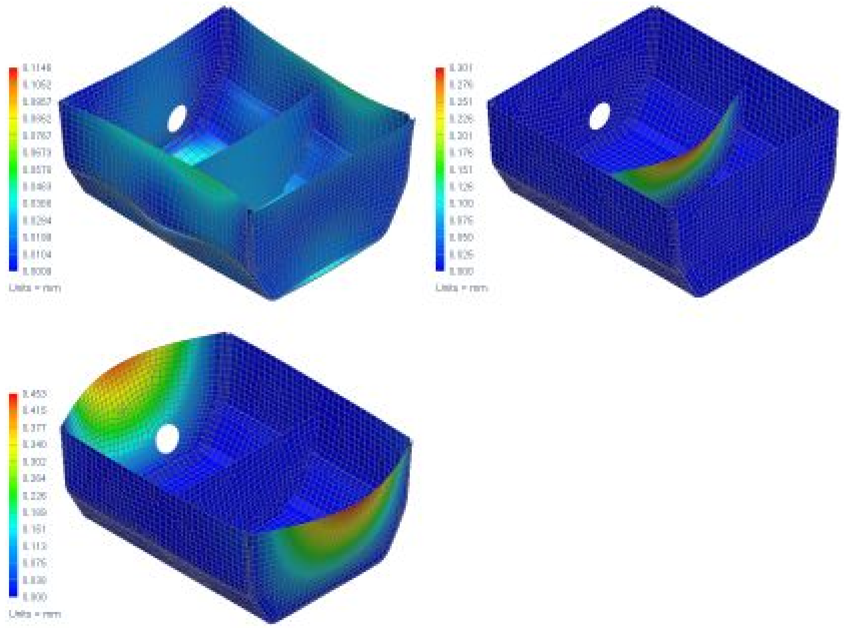
You can also apply a scale factor to the standard font size. In the **Post View** dialog box, on the **Annotation** tab, specify a **Scale Factor** between **0.5** and **2.0**.

Select and edit multiple post views

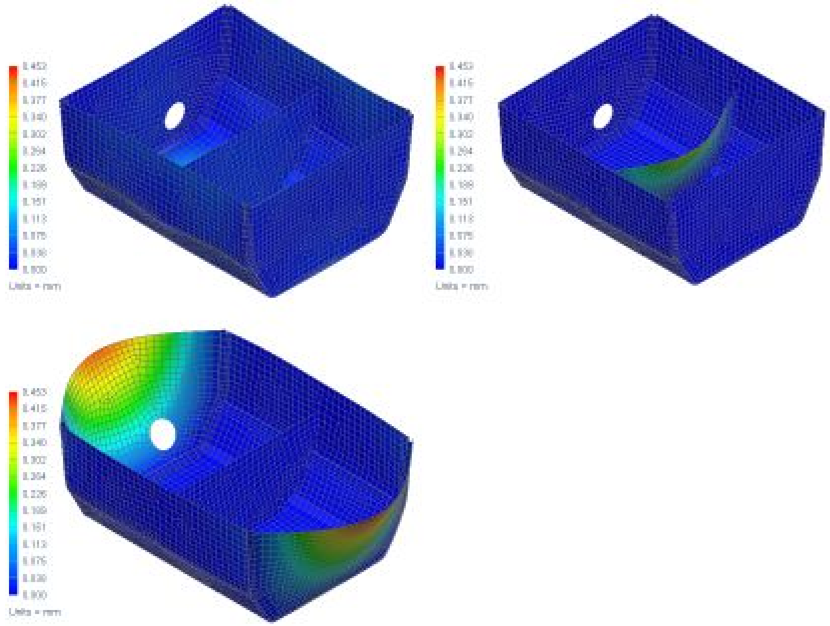
You can now apply post view settings to multiple post views simultaneously. The post views may be overlaid in a single viewport, or in separate viewports.

In the **Post-Processing Navigator**, use Shift-click or Ctrl-click to select multiple post views. You can then use the commands in the **Post-Processing** toolbar to apply the same settings to all post views simultaneously. For example, you can quickly apply consistent deformation scaling to all selected post views.

Use the new  **Get min/max for all selected postviews** button on the **Legend** tab of the **Post View** dialog box to scale to provide a consistent color bar scale across all displayed post views.



A multiple viewport display showing displacements from three different static load cases for the same model. The color bar and deformation are scaled independently for each post view.



Consistent color bar and deformation scaling applied to all three load cases. The relative displacements for the three load cases are more clearly displayed.

If you have multiple post views overlaid in the same viewport, you can use **Combined Viewport Result** and **Combined Viewport Displayed** to automatically scale the color bar across all post views overlaid in the current viewport.

Why should I use it?

Consistent, scalable fonts in post-processing ensures the readability of post view legends regardless of the size of the display. This is especially useful when capturing images for reports or presentations.

The ability to select and edit multiple post views is useful whenever you are overlaying or comparing results. It is especially convenient when post-processing external superelement system models containing a large number of superelement results sets. For more information, see **Unsatisfied xref title**.

Where do I find it?

Changing the font scaling:

Application	Advanced Simulation
Prerequisite	Analysis results loaded; one or more post views displayed.
Toolbar	**Post-Processing** ® **Edit Post View**

Location in dialog box	**Annotation** tab ® **Automatic Font Scaling** ® **Scale Factor**

Applying a single color bar scale to multiple post views:

Application	Advanced Simulation
Prerequisite	Analysis results loaded. Multiple post views displayed in one or more viewports. Multiple post views selected in the **Simulation Navigator**.
Toolbar	**Post-Processing** ® **Edit Post View**
Location in dialog box	**Legend** tab ® **Color and Value Control** group ® **Specified** ® **Get min/max for all selected postviews** **Legend** tab ® **Color and Value Control** group ® **Combined Viewport Result** **Legend** tab ® **Color and Value Control** group ® **Combined Viewport Displayed**

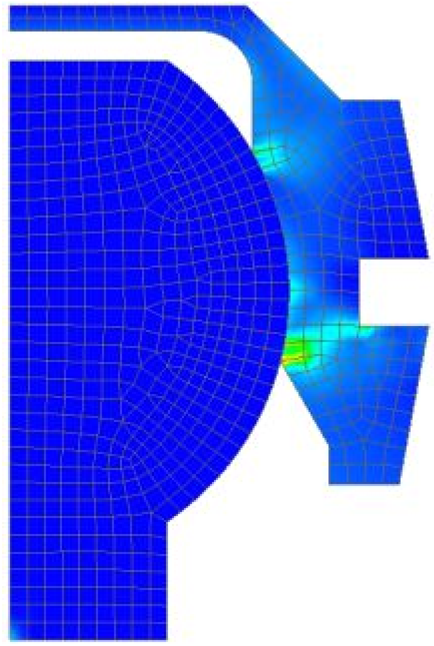
3D axisymmetric displays

What is it?

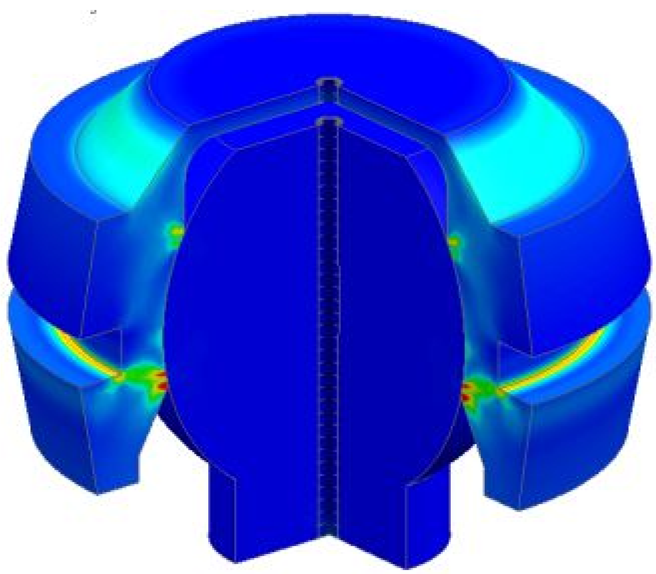
Use the **3D Axi-symmetric** option on the **Post View** dialog box to render axisymmetric results as a virtual solid model.

- You can set the angle of resolution. Set the **Revolve Angle** to 360 to render the full 3D part. Specify a number smaller than 360 to render the model as a cutaway, wedge, and so on.

- You can specify the number of sections to generate for the rendering.



Axisymmetric results displayed on free element faces.



The same results as a 270-degree 3D axisymmetric cutaway display. Edges are set to Feature.

Why should I use it?

You can generate high-quality, realistic 3D renderings of 2D axisymmetric results. Such displays enhance reports and presentation, and can improve communication with colleagues.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved model using axisymmetric elements.
Toolbar	**Post-Processing** ® **Edit Post View**
Location in dialog box	**Display** tab ® **Display on** ® **3D Axi-symmetric**

Export a result quantity to an expression

What is it?

After solving your model, you can now identify specific result quantities that are exported to named expressions. The maximum result value is exported to the expression. You can then use the expression to validate model requirements.

For example, you can create an elemental stress result measure for your solution for the von Mises component from subcase 1. An expression is created that contains the maximum value of von Mises stress from subcase 1. In the **Requirements Validation** dialog box, you could then associate the new expression with a stress requirement. For more information about associating an expression with a requirement, see the *Requirements Validation* online Help.

You can manage defined result measures in the **Result Measure Manager** dialog box. If you change the model and re-solve it, you must update the result measure in the **Result Measure Manager** dialog box.

Note If you solve the model with the **Run Solve in Foreground** option selected in the solution attributes, the result measure updates automatically.

For steps, see Create a result measure.

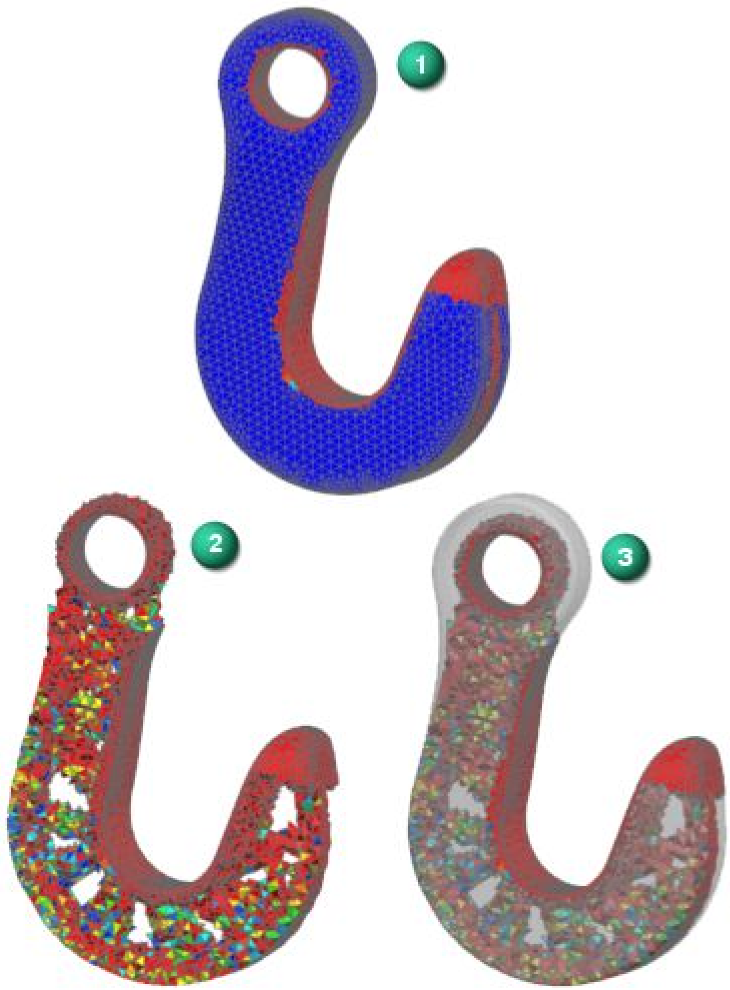
Where do I find it?

Toolbar	**Advanced Simulation** toolbar® **Result Measures**
Menu	**Insert**® **Result Measures**
Simulation Navigator	Right-click the simulation file® **Result Measures**

New options for results overflow/underflow

What is it?

In previous releases, you could specify a color to represent results that fall outside a specified range. In addition to this shaded display, you can now specify that overflow or underflow results values are shown as **Translucent** or **Clipped** (hidden).



(1) Material density results from a topology optimization. (2) Underflow results (below 0.05) set to Clipped. (3) Underflow set to Translucent.

Why should I use it?

Use **Clipped** to hide parts of your model where the results fall outside a specified range. Use **Translucent** to view only results within a specified range in the context of the full model.

These options are especially useful when viewing material density results from a topology optimization, to help you visualize the smoothed model.

Where do I find it?

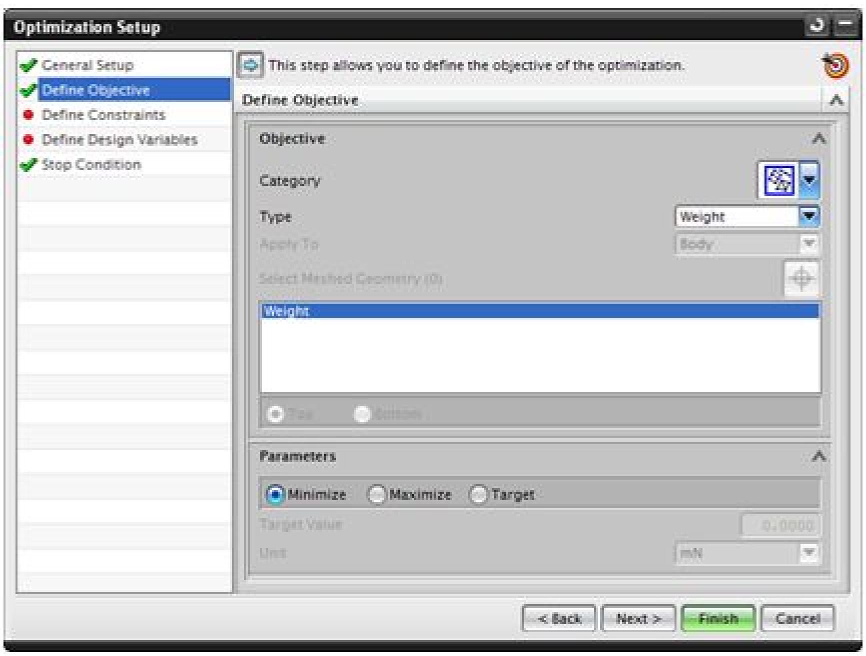
Application	Advanced Simulation
Prerequisite	Analysis results loaded; one or more post views displayed.
Toolbar	**Post-Processing** ® **Edit Post View**
Location in dialog box	**Legend** tab ® **Color and Value Control** group ® **Specified** ® **Overflow**. **Legend** tab ® **Color and Value Control** group ® **Specified** ® **Underflow**.

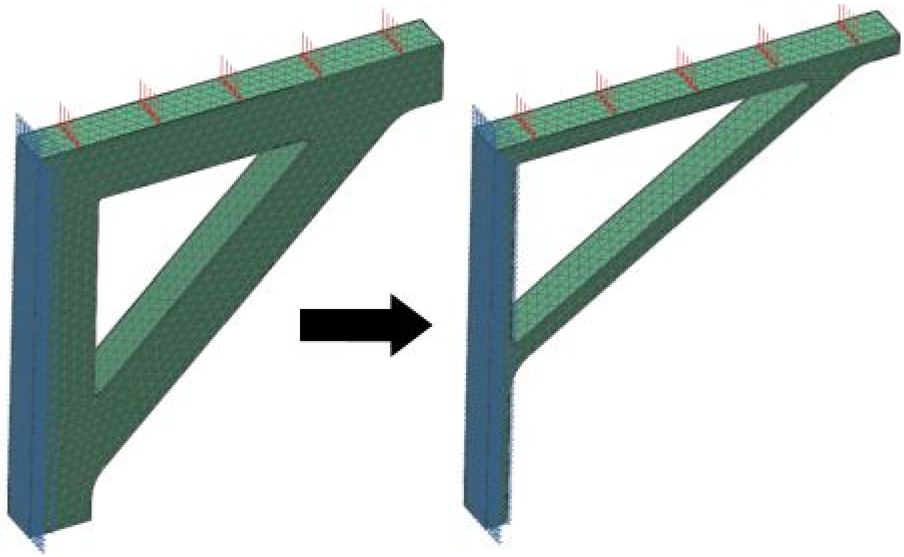
Optimization

Geometry Optimization update

What is it?

The **Optimization** product has been rewritten in the latest NX architecture, using a wizard-like user interface that guides you through the definition of the optimization problem. The optimization algorithm has also been improved to provide a greater degree of accuracy.





Where do I find it?

Application	Design Simulation, Advanced Simulation
Menu	**Insert® Geometry Optimization**
Simulation Navigator	Right-click the simulation® **New Solution Process® Geometry Optimization**

NX Topology Optimization

What is it?

This release introduces the ability to define a topology optimization using the Tosca Topology Optimization solver by FE Design.

NX Topology Optimization can help you to design a new component by providing you with an optimal design suggestion. The software generates this suggestion based on analysis results from your Advanced Simulation finite element model and the solution load cases that are solved in NX Nastran.

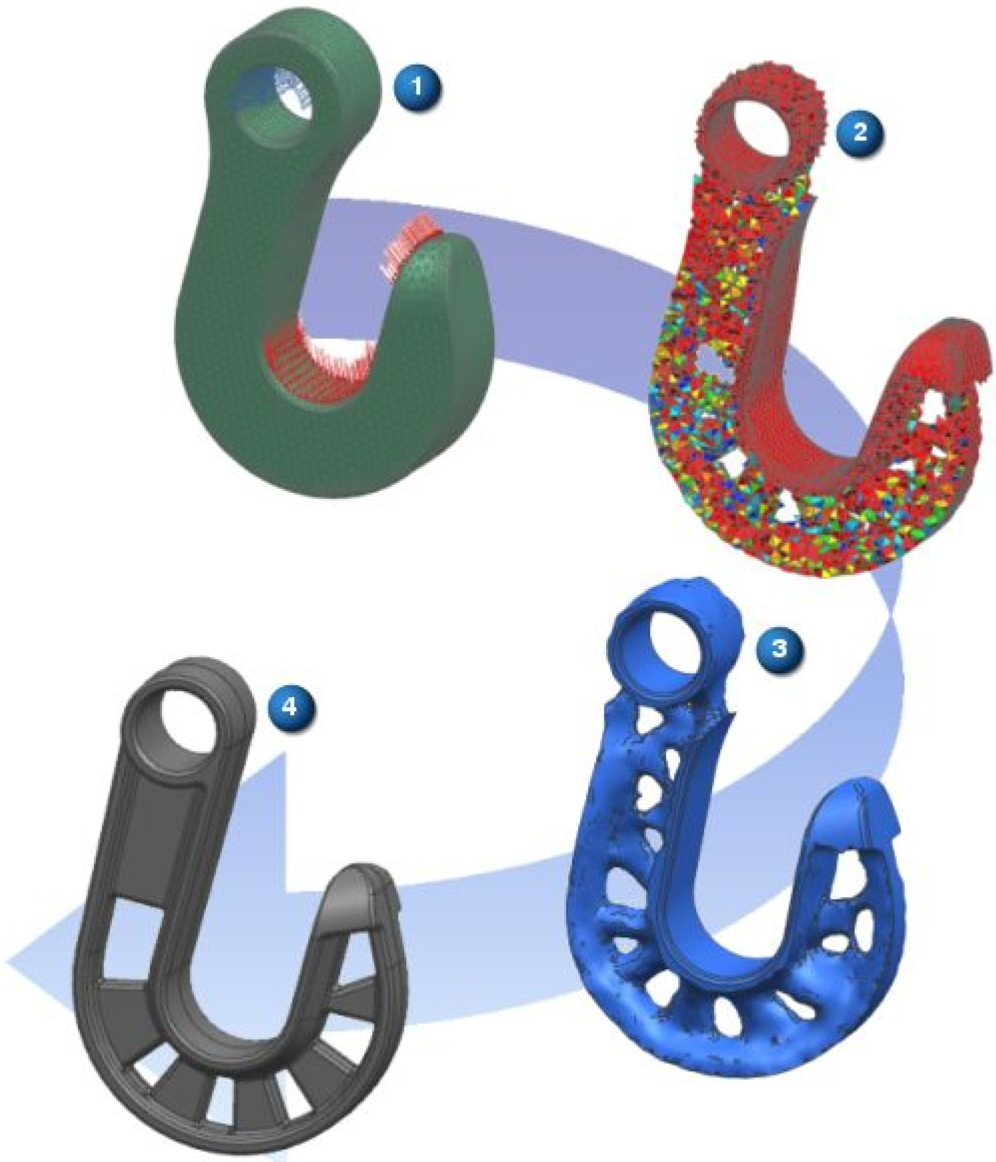
To set up the topology optimization, you define the following:

- A design area, which specifies the elements in the model that the optimization can modify.

- A design objective, such as maximizing stiffness.

You can define design constraints, such as volume, and also manufacturing constraints to ensure that the component can be produced. Working within these bounds, the optimization determines an optimal material distribution by adjusting the material density of the elements in the design area.

The optimization assigns to each element a relative density between 0 and 1. Elements with densities closer to 0 are soft elements, and elements with densities closer to 1 are hard elements. The resulting model has a block-like material distribution. The software then creates transitions to smooth the model, which result in a more continuous distribution of material. You can import this smoothed model into NX Modeling to use as a starting point for creating your new component.



(1) Design area for optimization; (2) optimized model; (3) smoothed model; (4) final CAD design

Supported solution types, design objectives, and constraints include:

Nastran Solution	Supported Design Objective	Supported Design Constraints
SESTATIC 101 – Single Constraint	Volume Stress/Strain (stiffness) Displacement	Volume Displacement Reaction Force
SEMODES 103	Volume Frequency	Volume Frequency

The following NX Nastran element types are supported:

- CHEXA, CTETRA, CPENTA
- CQUAD, CQUAD4, CQUAD8, CQUADR, CSHEAR, CQUADX
- CTRIA3, CTRIA6, CTRIAR, CTRIAX, CTRAX6
- CBAR, CBEAM, CBEND, CBUSH, CGAP, CONM1, CMASS1

The following limitations apply:

- With SESTATIC 101 solutions, you can constrain a maximum of 5 nodes with your optimization design constraints. For example, you can have 5 design constraints that each specify a single node, or you can have 1 design constraint that specifies 5 nodes, or any such combination.
- Only the standard SEMODES 103 solution is supported. Response Simulation, Flexible Body Analysis, and Superelement solutions are not supported.
- With SEMODES 103 solutions, you can include only one subcase in the optimization.
- **Enforced Displacement** constraints and **Temperature** loads are not supported.
- **Displacement** design constraints can only be defined at nodes to which loads have been applied.
- **Reaction Force** design constraints can only be defined at nodes that are constrained with an SPC constraint.

For steps, see NX Topology Optimization workflow.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Nastran SESTATIC 101 or SEMODES 103 solution
Menu	**Insert® Topology Optimization**
Simulation Navigator	Right-click the simulation® **New Solution Process® Topology Optimization**

Durability

Laminate support for Durability

What is it?

Durability now supports stress and strain results from models with defined laminate physical property tables. NX calculates durability results from the top and bottom ply stress or strain results.

If your laminate has at least one non-isotropic material, you must override the material by an isotropic material in the **Static/Transient Durability Event** dialog box. In that case, all plies have the same isotropic material.

Durability does not support the following laminates:

- Laminates that have plies with material type set to **Ply Materials**.
- Laminates that are defined by using global layups.
- Laminates that are defined by using the solid laminate physical property table.

For more information, see Supported solvers and solutions for durability analysis.

Analyzing strain gage rosette data

What is it?

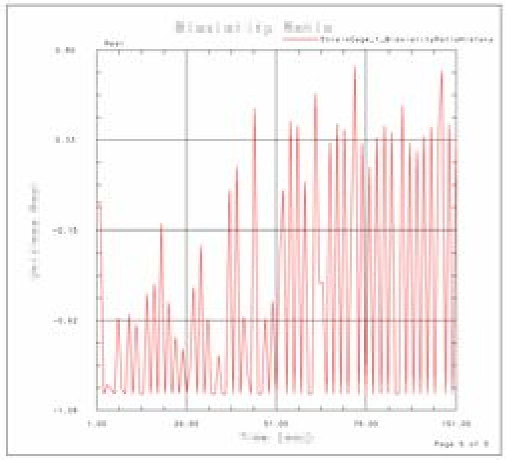
Use the new **Analyze Strain Gage** command to create durability functions from strain gage rosette data. You can obtain the strain gage rosette data from either of the following:

- A Response Simulation solution process. You can select the strain gage data from a list or in the graphics window.
- An AFU file. You can obtain the strain gage data stored in the AFU file from a test setup.

The durability functions are saved to a target AFU file. You can view the durability functions by using the **XY Function Navigator**.

The following durability functions can be obtained:

- Biaxiality ratio history
- Effective stress and strain as functions of time
- Maximum and absolute maximum principal strain as functions of time
- Maximum and absolute maximum shear strain as functions of time
- Maximum and absolute maximum principal stress as functions of time
- Maximum and absolute maximum shear stress as functions of time
- Von Mises and signed Von Mises stress as functions of time
- Non-proportionality indicator functions: maximum and absolute maximum principal angle as functions of time



For more information, see Analyzing strain gage rosette data.

Why should I use it?

The **Analyze Strain Gage** command calculates and stores stress and strain functions for a specified strain gage. You can produce a spreadsheet that contains damage information for the strain gage location, by using the **Evaluate Damage** command.

The **Analyze Strain Gage** command also provides biaxiality ratio and non-proportionality indicators that can help you decide which fatigue life criteria to use for the damage evaluation.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability ® Analyze Strain Gage**
Menu	**Insert ® Durability ® Analyze Strain Gage**

Durability damage evaluation

What is it?

Use the new **Evaluate Damage** command to perform cycle counting to identify all stress or strain cycles in a duty cycle and to calculate the damage due to identified cycles. This command uses stress or strain functions that are saved in AFU files. The AFU files must be loaded in the **XY Function Navigator**.

The fatigue damage histogram results can be stored either in a spreadsheet or a CSV file.

You must specify the following:

- One or more stress or strain functions. NX cumulates fatigue damage results from all specified functions.
- Nominal range bin definition and mean bin definition. These definitions are automatically calculated from the specified functions, but you can modify their values.
- Fatigue life criterion. All fatigue life criterion available for the durability events are available, including the new **Title not found**.
- The cyclic stress-strain model.
- The material to which the specified stress or strain functions are applied.

The stress and strain functions can be obtained from strain gage data using the new **Analyze Strain Gage** command.

For more information, see Durability damage evaluation.

Why should I use it?

When you use the **Evaluate Damage** command, you can compute damage at a location without having an FE model with stress or strain results. The durability damage evaluation is an alternative to the event-based fatigue damage evaluation.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Evaluate Damage**
Menu	**Insert** ® **Durability** ® **Evaluate Damage**

Durability damage report

What is it?

Use the new **Report Durability Damage** command to export fatigue damage results from the durability solution process to the following output formats:

- An Excel spreadsheet using the **Export to Spreadsheet** option.

- A CSV file using the **Export to CSV File** option.

These options are available only if the durability results node exists in the **Simulation Navigator**, and if you select the **Event Damage** check box for at least one event. The **Event Damage** check box is located on the **Fatigue** tab of the **Static/Transient Durability Event** dialog box.

For more information, see Durability results for the event-based analysis.

Why should I use it?

You can efficiently post process fatigue damage data that is computed by NX Advanced Durability.

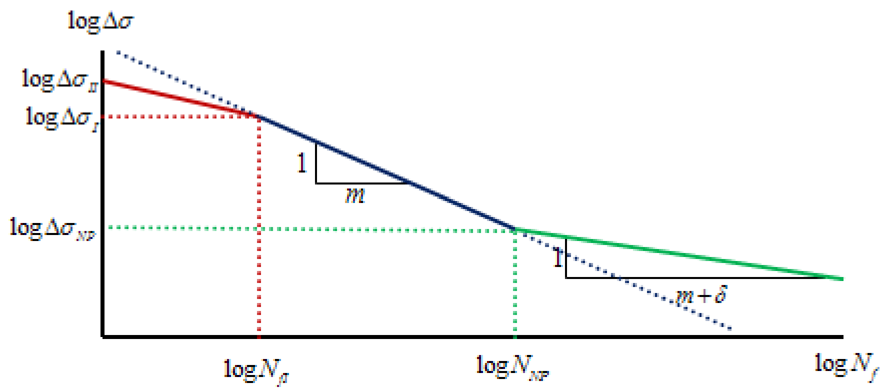
Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Report Durability Damage**
Menu	**Insert** ® **Durability** ® **Solution** ® **Report Damage**
Simulation Navigator	Right-click the Durability solution process node ® **Report Damage**

TWI fatigue life criterion

What is it?

The **TWI** fatigue life criterion extends the BWI fatigue life criterion into the low cycle and high cycle fatigue regimes.



Schematic of TWI S-N curve: the blue curve is the BWI S-N curve, the red curve is the TWI extension into the low cycle, and the green curve is the TWI extension into the high cycle

For the TWI life criterion, you need to specify the same parameters that you specified for the BWI life criterion:

- The weld class.
- The number of standard deviations below the mean value of the stress range or the probability of failure.

In addition, you also need to specify the following unique TWI parameters:

- For the extension of the BWI criterion into the high cycle region, you must specify the non-propagating stress range, N_{NP}, and its slope change, d.
- For the extension of the BWI criterion into the low cycle region, you must specify the low cycle stress range cutoff, Δs_I, and its extension, Δs_{II}, as a proportion of the yield stress.

For more information, see TWI fatigue life criterion.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Static Durability Event** or **Transient Durability Event**
Menu	**Insert** ® **Durability** ® **Event** ® **Static** or **Transient**
Simulation Navigator	Right-click the Durability solution process node ® **New Event** ® **Static** or **Transient** Right-click the static event or transient event node ® **Edit**
Location in dialog box	**Fatigue** tab ® **Fatigue Life** group ® **Fatigue Life Criterion** list

BWI fatigue life criterion enhancement

What is it?

For the BWI fatigue life criterion, you can now set the probability of failure value instead of the number of standard deviations. NX converts the probability of failure value into the number of standard deviations below the mean of the BWI S-N curve.

The probability of failure, P_f, and the number of standard deviations, d, are related by the following equation:

$$P_f = 1 - \frac{1}{2}\mathrm{erf}\left(\frac{d}{\sqrt{2}}\right) \times 100\%$$

The *erf* function is the Gauss error function that is defined as:

$$\mathrm{erf}(x) = \frac{2}{\sqrt{\pi}} \int_0^x e^{-t^2} dt$$

For more information, see BWI fatigue life criterion.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Static Durability Event** or **Transient Durability Event**
Menu	**Insert** ® **Durability** ® **Event** ® **Static** or **Transient**
Simulation Navigator	Right-click the Durability solution process node ® **New Event** ® **Static** or **Transient** Right-click the static event or transient event node ® **Edit**
Location in dialog box	**Fatigue** tab ® **Fatigue Life** group ® **Fatigue Life Criterion** list ® **BWI** ® **Weld Data** subgroup ® **Option** list

I-deas durability material library

What is it?

You can now use materials for which durability properties are predefined. These materials come from the I-deas durability material library.

For more information, see Material properties for durability analysis.

Where do I find it?

Application	All
Menu	**Tools** ® **Material Library Manager**
Toolbar	**Advanced Simulation** toolbar ® **Manage Materials**
Location in dialog box	Select the **Site MatML Library** or **User MatML Library** checkbox ® **Browse** ® [NX_installation]/NXCAE_EXTRAS/advancedduarbility/Advanced

Performance enhancement when solving transient events

What is it?

NX Advanced Durability can now calculate the durability results from the stress or strain results that are saved in the SORT2 format in the OP2 file.

The access to SORT2 stress and strain data is available only for NX Nastran SOL 109 and SOL 112 solutions.

Note The OP2 file size must be less than 1 GB.

Why should I use it?

When you solve transient events for durability results, accessing SORT2 stress and strain data could improve performance for solutions with a large number of iterations.

Where do I find it?

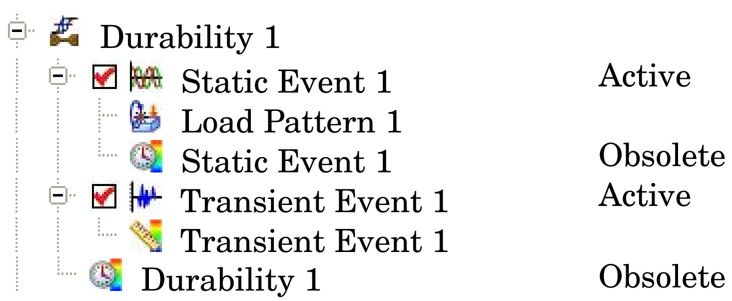
Application	Advanced Simulation
Prerequisite	A transient structural solution
Toolbar	**Durability ® Transient Durability Event**
Menu	**Insert ® Durability ® Event ® Transient**
Simulation Navigator	Right-click the Durability solution process node ® **New Event ® Transient**
Location in dialog box	**Use SORT2 Stress/Strain Data**

Obsolete status for the event result node

What is it?

When a durability event is modified, its node is now flagged as obsolete in the **Simulation Navigator**. The obsolete results icon appears in front of the event result node.

In previous releases, the cumulated durability result node was flagged as obsolete only if you added an event or deactivated an event whose results are cumulated. Now, it is also flagged when an event whose results are cumulated is modified.



For more information, see Simulation Navigator for Durability.

Why should I use it?

You can quickly detect durability events with outdated settings.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A durability solution process with a solved event

Durability analysis in the Modeling application

What is it?

The **Durability Wizard** is now available from the Modeling application. To run the **Durability Wizard**, you need to have a Simulation file that contains stress or strain results from a static stress analysis solution. To obtain this Simulation file, use the **Stress Wizard**.

Why should I use it?

You can test your parts for fatigue damage directly in the Modeling application by doing a durability analysis.

Where do I find it?

Durability Wizard

Application	Modeling, Design Simulation, Advanced Simulation
Prerequisite	Results from a static stress analysis solution
Menu	**Tools** ® **Durability Wizard**

Stress Wizard

Application	Modeling, Design Simulation, Advanced Simulation
Resource bar	**Process Studio** ® **NX CAE Stress Wizard**

NX FE Model Correlation and NX FE Model Updating

Exciters in pre-test solution processes

What is it?

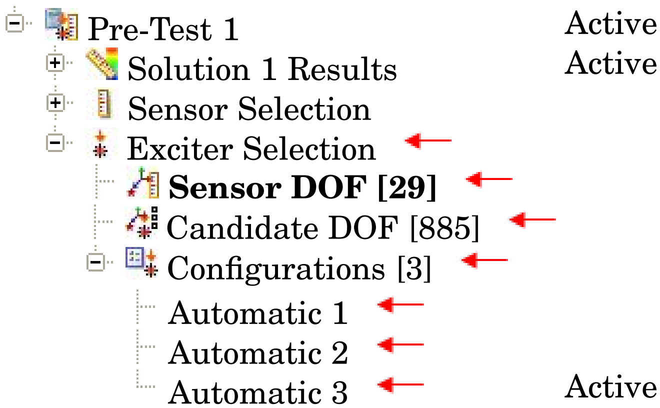
You can now select exciters when you create a pre-test solution process. You must specify a sensor DOF set, a candidate DOF set, and the settings for the exciter selection configuration. NX uses an exciter selection algorithm to find the optimum number of exciters and the optimum location for these exciters.

- Use the new **Edit Sensor DOF for Exciters** command to define the sensor DOF set that contains the sensor locations with appropriate sensor translational DOFs. You can obtain the sensor DOF set from the sensor selection configuration, or you can manually choose sensor DOFs.

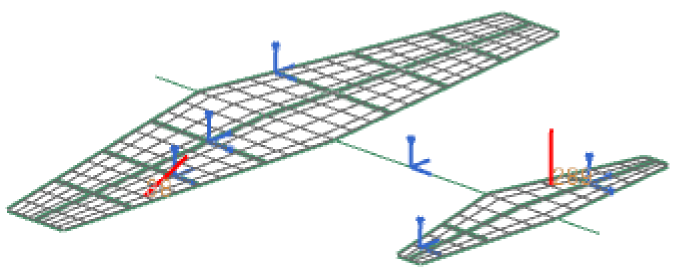
- Use the new **Edit Candidate DOF for Exciters** command to define the candidate DOF set that contains the possible location for the exciters.

- Use the new **New Exciter Configuration** command to specify the settings for the exciter selection algorithm.

When you create a pre-test solution process, the exciter selection nodes are created automatically in the **Simulation Navigator**. The **Configurations [#]** node can contain multiple exciter selection configurations that you can solve either simultaneously or one by one. The new exciter selection nodes are shown with a red arrow in the following graphic.



In the graphics window, the exciter locations are shown in red and the sensor DOFs in blue.



For more information, see Exciter selection configuration.

Why should I use it?

NX now allows you to obtain optimal number of exciters and exciter locations before performing your modal testing in a testing facility. Previously, this was not possible in NX.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A pre-test solution process must exist.
Toolbar	**Pre-Test Planning** ® **Edit Sensor DOF for Exciters**  or **Edit Candidate DOF for Exciters**  or **New Exciter Configuration** 
Menu	(Sensor DOF set) **Tools** ® **Pre-Test Planning** ® **Exciter** ® **Edit Sensor DOF** (Candidate DOF set) **Tools** ® **Pre-Test Planning** ® **Exciter** ® **Edit Candidate DOF** (Exciter Configuration) **Tools** ® **Pre-Test Planning** ® **Exciter Configurations** ® **New**
Simulation Navigator	(Sensor DOF set) **Exciter Selection** node ® right-click the **Sensor DOF [#]** node ® **Edit** (Candidate DOF set) **Exciter Selection** node ® right-click the **Candidate DOF [#]** node ® **Edit** (Exciter Configuration) **Exciter Selection** node ® right-click the **Configurations [#]** node ® **New**

Exciter configuration commands

What is it?

Use the new exciter configuration commands to create, edit, or solve exciter selection configurations.

- Use the **New Exciter Configuration** command to create a new exciter selection configuration. The new configuration node is created in the **Simulation Navigator** under the **Configurations [#]** node. The updated icon  appears in front of the configuration node to indicate that you need to solve it.

- Use the **Edit Active Exciter Configuration** command to edit the active exciter selection configuration. You can modify the damping ratio, the threshold for the normal mode indicator function (NMIF), the off-axis angles for the exciter direction, and the name of the exciter selection configuration.

- Use the **Solve Active Exciter Configuration** command to solve the active exciter selection configuration for the optimal number of exciters and exciter locations.

- Use the **Solve All Exciter Configurations** command to simultaneously solve all exciter selection configurations when you select or deselect the solution's normal modes, or when you edit either the sensor DOF set or the candidate DOF set.

- Use the **Information** right-click command on an exciter selection configuration to display the details for an exciter set. Details can include the number of excited modes in the configuration, and the modes that are excited by each exciter.

- Use the **Information** right-click command on the **Configurations [#]** node to display the exciter selection settings for all the configurations.

For more information, see Exciter selection configuration.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A pre-test solution process must exist.

Creating or editing a configuration

Toolbar	**Pre-Test Planning** ® **New Exciter Configuration** or **Edit Active Exciter Configuration**
Menu	**Tools** ® **Pre-Test Planning** ® **Exciter Configurations** ® **New** or **Edit Active**
Simulation Navigator	(Create configuration) **Exciter Selection** node ® right-click the **Configurations [#]** node ® **New** (Edit configuration) **Exciter Selection** node ® **Configurations [#]** node ® right-click an Automatic node ® **Edit**

Solving configurations

Toolbar	**Pre-Test Planning** ® **Solve Active Exciter Configuration**  or **Solve All Exciter Configurations** 
Menu	**Tools** ® **Pre-Test Planning** ® **Solve for Exciters** ® **Active** or **All**
Simulation Navigator	**Exciter Selection** node ® **Configurations [#]** node ® right-click an Automatic node ® **Solve** **Exciter Selection** node ® right-click the **Configurations [#]** node ® **Solve All**

Getting exciter selection configuration information

Simulation Navigator	**Exciter Selection** node ® **Configurations [#]** node ® right-click an Automatic node ® **Information** **Exciter Selection** node ® right-click the **Configurations [#]** node ® **Information**

New supported design variables for NX FE Model Updating

What is it?

The DESOPT 200 — Model Update solution now supports the following new design variable types:

- **PCOMP**
- **Laminate**

The following table shows which physical fields can be optimized for these two design variables.

Physical Field (PNAME)	**Description**
Z0	Distance from the reference plane to the bottom surface
NSM	Non-structural mass per unit area
SB	Allowable interlaminar shear stress
TREF	Reference temperature
GE	Damping coefficient
Ti	Thickness of ply *i*
THETAi	Orientation angle of the longitudinal direction of ply *i* with the material axis of the element.

For each **PCOMP** or **Laminate** design variable, you can optimize the thickness and the orientation angle for one or more plies.

For more information, see Supported design variables for Model Update solution process.

Why should I use it?

NX FE Model Updating allows the definition of design variables for models with composite materials.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Modeling Objects** ® **Design Variable — Composite Property** ® **Create**
Menu	**Insert** ® **Modeling Objects** ® **Design Variable — Composite Property** ® **Create**

NX Laminate Composites

Import layup using lamination shorthand format

What is it?

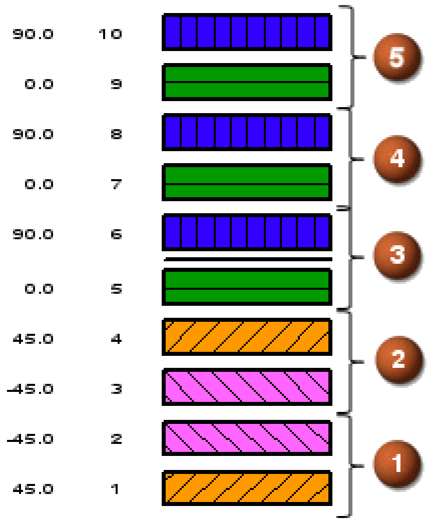
Use the new **Import Layup Using Shorthand Format** option to quickly define a layup using the lamination shorthand format. You can specify a thickness and a material for each ply.

You enter the shorthand notation in the **Shorthand String** box in the **Shorthand Format** dialog box. The shorthand notation has the following conventions:

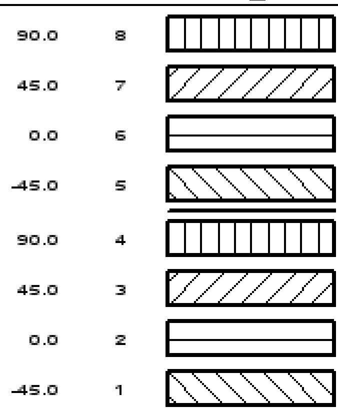
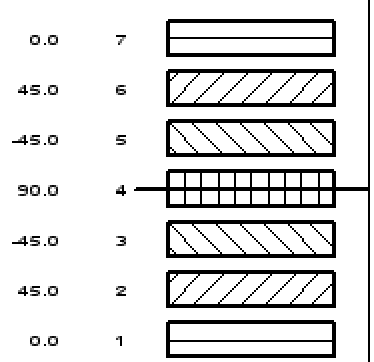
- A ply layup is contained inside [] (square brackets).
- Ply angles are separated by / (forward slash).
- Ply groups are contained inside () (parentheses).
- Symmetry is indicated by the suffix _s (underscore and *s*). For a symmetric layup, the suffix is placed after the square brackets. For a symmetric group, the suffix is placed after the parentheses.
- The core ply is indicated by ‘ (apostrophe).
- A repeated layup, group, or ply is indicated by _# (underscore and number sign) where # indicates the number of repetitions.

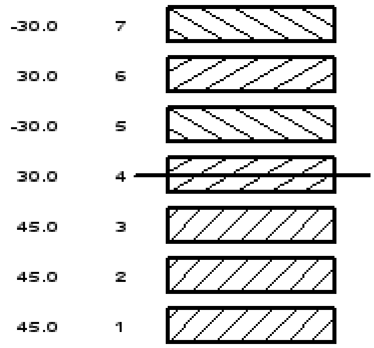
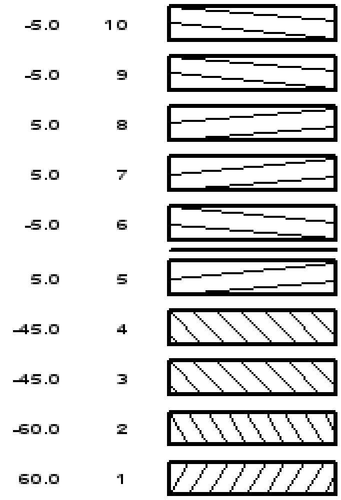
- + (plus) and – (minus) in front of a number indicate the positive and negative angles.

Example When you enter *[(+-45)_s(0/90)_3]*, your ply layup is composed of 10 plies and 5 groups as shown in the following graphic. Group 2 is a symmetric repeat of group 1. Groups 4 and 5 are non-symmetric repeats of group 3.



The following table shows you additional examples of the shorthand format.

Shorthand format	[-45/0/45/90]_2	[0/+-45/90']_s
Ply layup with angles and ply numbers		
Shorthand format	[45_3/+-30_2]	[+-60/- -45/+-++- -5]

Ply layup with angles and ply numbers		

For more information, see Defining a layup using the lamination shorthand format.

Why should I use it?

This command lets you create complicated ply layups quickly and efficiently.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Toolbar	**Laminates** ® **Laminate Physical Property** or **Global Layup**
Menu	**Insert** ® **Laminate** ® **Physical Property** or **Global Layup**
Location in the dialog box	**Ply Layup** group ® **Import Layup Using Shorthand Format**

Solid Laminate physical property

What is it?

In the Nastran environment, you can now create a **Solid Laminate** physical property that is attached to a 3D mesh collector. The **Solid Laminate** physical property is the 3D equivalent of the **Laminate** physical property.

You can use the **Solid Laminate Modeler** dialog box to:

- Create plies and ply groups. Use the new **Import Layup Using Shorthand Format** option to quickly define a layup using the lamination shorthand format.
- Define a ply stacking sequence and a stacking recipe.
- Define laminate-specific and solver-specific options.

For each ply, you must specify the following parameters:

- Material
- Thickness
- Angle
- Ply failure theory

You can also specify the inter-laminar failure theory separately between consecutive plies.

The solid laminate is exported to the PCOMPS bulk data entry card.

The 3D mesh must consist of either CHEXA or CPENTA elements.

For more information, see Creating a laminate physical property.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM within the Nastran solver environment.
Toolbar	**Advanced Simulation** ® **Physical Properties**
Menu	**Insert** ® **Physical Properties**
Simulation Navigator	Right-click a 3D collector node ® **Edit**
Location in dialog box	(**Physical Properties** command) **Type** list ® **Solid Laminate** ® **Create** (Right-click command) **Type** list ® **Solid Laminate** ® **Create Physical**

Nastran support for Extrude Laminate and Fill Laminate commands

What is it?

The **Extrude Laminate** and **Fill Laminate** commands are now supported in the Nastran environment.

Only CQUAD8 and CTRIA6 2D Nastran elements are supported. The extruded 3D mesh consists of either CHEXA or CPENTA elements, and the ply drop-offs consist of CPENTA, CTETRA, or CPYRAM elements.

For both commands, NX creates a 3D mesh collector that has a **Solid Laminate** physical property assigned to it. NX sets the **Stacking Recipe** in the **Solid Laminate Modeler** dialog box to **Extruded**.

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.

The 3D mesh collector contains one mesh per ply. At solve time, each mesh is exported using a PCOMPS card that contains a single ply.

For more information, see Inflating laminates.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM within the Nastran solver environment.
Toolbar	**Laminates** ® **Extrude Laminate**  or **Fill Laminate** 
Menu	**Insert** ® **Laminate** ® **Extrude** or **Fill**

Material orientation of extruded and filled 3D elements

What is it?

NX Laminate Composites now exposes material orientation vectors for extruded elements in both Nastran and ANSYS environments.

For more information, see View material orientation graphically.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Finite Element Model Check**
Simulation Navigator	Right-click a 3D extruded mesh node ® **Check** ® **Material Orientation**
Location in the dialog box	**Element Material Orientation**

Exporting laminate physical properties to FiberSIM

What is it?

Use the **Export to FiberSim** command to export laminate physical properties to the FiberSIM PDI format in either of the following files:

- An Excel spreadsheet
- A tab separated text file

Note The layups are not exported when you use this command. To export layups, you must use the **Export Plies to FiberSim** command.

For more information, see Creating a laminate physical property.

Why should I use it?

Because the Excel spreadsheet and the tab separated text file follow the FiberSIM PDI format, you can import the laminate to the FiberSIM application and apply the simulation laminate to CAD geometry.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Menu	**Insert** ® **Laminate** ® **Export to FiberSim**

Exporting global layup to FiberSIM

What is it?

Use the **Export Plies to FiberSim** command to export global plies in a global layup to FiberSIM using the HDF5 format. *HDF5* is a general use hierarchical data format designed to store large amounts of numerical data. The exported file has the .h5 extension.

The exported FiberSIM HDF5 file contains the ply stacking sequence and the following data for each ply:

- The ply number
- The material name
- The ply angle
- A coordinate system, which is located at the starting point of the draping and whose X-axis is aligned with the 0° alignment of the fibers. For plies that are created using the projection draping solver or that are imported from a FiberSIM XML file, the starting point and the direction of the draping are created at export. Each coordinate system becomes a separate Rosette in FiberSIM.
- The mesh that contains node coordinates. All stored elements have either three nodes or four nodes. FiberSIM computes a ply boundary from the mesh.

The ply thickness is not exported. FiberSIM extracts the ply thickness from the material definition.

Note Laminate physical properties are not exported when you use this command. To export laminate physical properties, you must use the **Export to FiberSim** command.

For more information, see Creating a global layup.

Why should I use it?

Because the global plies are stored in the FiberSIM HDF5 file, you can import them to the FiberSIM application.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Simulation Navigator	Right-click a global layup node ® **Export Plies to FiberSim**

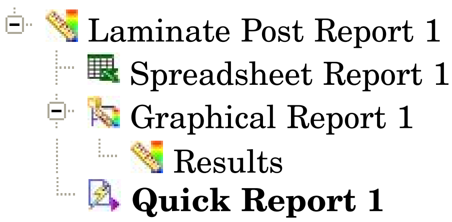
Laminate post processing enhancements

What is it?

A new **Advanced Post Report** command is available on the **Advanced Simulation** toolbar. This command unifies and replaces the following three commands:

- **Spreadsheet Post Report**
- **Graphical Post Report**
- **Quick Post Report**

Within the current Simulation file, the **Advanced Post Report** command lets you select one or more results sets with a specific solution, loadcase, and iteration. The result sets can include imported results. After you select the result sets, a laminate post report node is created in the **Simulation Navigator**. From the laminate post report node, you can create the spreadsheet report, the graphical report, or the quick report.



The following commands appear when you right-click the laminate post report node in the **Simulation Navigator**.

Icon	Command name	Description
	Edit	Lets you modify the input selection for the laminate post report metasolution.
	Clone	Clones the laminate post report metasolution and all its reports.
	Delete	Deletes the laminate post report node and the attached reports.
	Create Spreadsheet Report	Creates a spreadsheet report node from which you can export the laminate results to an Excel spreadsheet or to a CSV file.
	Create Graphical Report	Creates a graphical report node from which you can generate laminate results that you can display in the **Post Processing Navigator**.
	Create Quick Report	Creates a quick report node from which you can export the laminate results to an Excel spreadsheet or to a CSV file.
	Information	Opens the **Information** window that lists the selected result sets, spreadsheet report settings, graphical report settings, and quick report settings.

	Make Active	Appears only if the laminate post report metasolution is not active. Activates the laminate post report metasolution.
	Make Inactive	Appears only if the laminate post report metasolution is active. Deactivates the laminate post report metasolution.

For more information, see Advanced post processing of laminate results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the Simulation file.
Toolbar	**Laminates** ® **Advanced Post Report**
Menu	**Insert** ® **Laminate** ® **Advanced Post Report**

Support for PSHELL in Advanced Post Report

What is it?

Laminate post processing now has the ability to compute ply stress for laminates that were exported using the PSHELL output format. Previously, you needed to change the output format from PSHELL to PCOMP in the **Laminate Modeler** dialog box.

Selecting disconnected faces and elements with the projection draping solver

What is it?

When you drape global plies using the projection draping solver, you can now select faces or 2D elements that are not contiguous.

In previous NX versions, the **Projection** option was called **None**.

For more information, see Draping solvers.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Simulation Navigator	Right-click one or more global ply nodes ® **Edit**
Location in the dialog box	**Solver** ® **Projection**

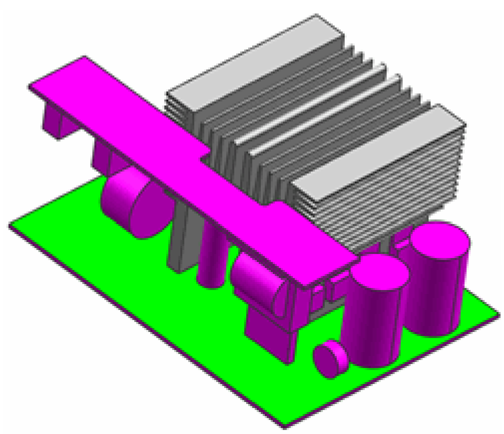
NX Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal

Surface Wrap Fluid Domain

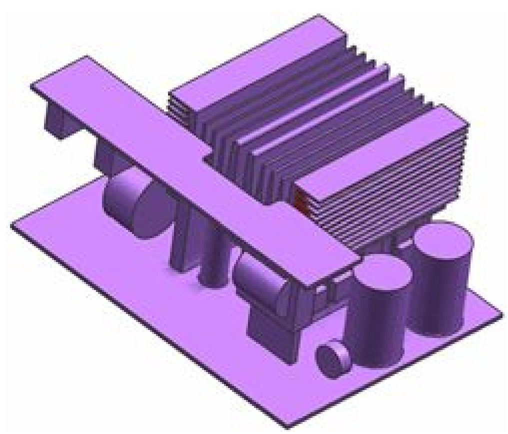
What is it?

Use the **Create Surface Wrap Fluid Domain** command to generate polygon fluid body geometry using a surface wrapping algorithm. The wrapping algorithm is based on the selected bodies that define its surface boundaries and embedded bodies. The created polygon fluid body geometry has the same characteristics as a regular NX polygon body.

The wrapping algorithm overlays an octree Cartesian triangular mesh onto the geometry and refines the octree mesh as it intersects the geometry until an octree mesh with a target resolution is obtained. An airtight envelope, called *surface wrapping*, is created by projecting nodes onto the geometry surfaces, edges, and points. The polygon fluid body geometry is generated using the surface wrapping.



Original geometry comprised of multiple polygon bodies



One fluid body wrapped around all the bodies

To create polygon fluid body geometry, you must:

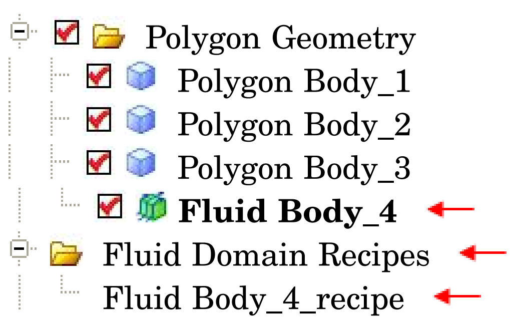
1. Select polygon bodies that define the fluid domain boundaries.
2. Specify whether you want to create an interior fluid domain or a fluid body that wraps around the exterior of all the selected bodies. You can create an interior fluid domain by specifying either an interior point or a face and the normal vector toward the interior region.
3. Specify a global surface wrapping resolution. The **Auto Calculate** option calculates a global resolution for you.

Note When you create an interior fluid domain body, if your original geometry has holes or cracks that are bigger than the global resolution, the wrapping algorithm stops with an error message and displays the leaking path. The leaking path is the path that links the internal point to the outside region that goes through the hole. To create the fluid body, you must close the holes or choose a bigger resolution.

In the **Surface Wrap Fluid Domain** dialog box, you can also select the following advanced options:

Snap to Source Edges	Makes the fluid body edges snap to the edges of the source polygon bodies.
Facet Decimation	Controls how planar surfaces are defined.

In the **Simulation Navigator**, NX creates the **Fluid Domain Recipes** container node, a fluid body recipe node, and a fluid body node under the **Polygon Geometry** container node.



The fluid body recipe node lets you access the **Surface Wrap Fluid Domain** dialog box which contains options of the specified fluid body.

NX updates the fluid body automatically whenever you make changes to any of the source polygon bodies or to the fluid body recipe.

In the **Customer Defaults** dialog box, you can control:

- The colors of the hole faces, regular faces, and stitch faces.
- The translucency of the fluid body.
- The factor that is used to compute the suggested value of global resolution when you click **Auto Calculate** in the **Surface Wrap Fluid Domain** dialog box.

For more information, see Surface Wrap Fluid Domain.

Why should I use it?

The **Create Surface Wrap Fluid Domain** command helps you to easily create polygon geometry for your fluid domain, such as the air volume of models that comprise many solid parts and complicated geometry.

Supported solvers and analysis types

Solver	Analysis Type
NX Electronic Systems Cooling	Coupled Thermal-Flow
NX Thermal and Flow	Flow Coupled Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Toolbar	**Fluid Domain** ® **Create Surface Wrap Fluid Domain**
Menu	**Insert** ® **Fluid Domain** ® **Create Surface Wrap Fluid Domain**
Simulation Navigator	Right-click the fluid body recipe node under **Fluid Domain Recipes** node ® **Edit**

Solving flow analysis in parallel

What is it?

You can now run flow analyses and coupled thermal and flow analyses in parallel on a single workstation using up to 8 processes with Advanced Flow and NX Electronic Systems Cooling products.

Note With the new NX Thermal and Flow DMP product, you can extend this parallel processing capability to multiple workstations and more than 8 processes.

The following basic boundary conditions are available:

- **Deactivation Set**
- **Flow Blockage**
- **Flow Boundary Condition**
- **Flow Surface**
- **Fluid Domain**
- **Report**
- **Screen**
- **Selective Results**
- **Symmetry Plane**
- **Thermal Loads**
- **Initial Conditions**
- **Mapping**
- **Temperature**

Steady-state and transient analyses are supported for forced and natural convection problems. Models with disjoint meshes are not supported.

For more information, see Parallel processing in thermal and flow analyses.

Why should I use it?

Use the parallel processing option of the flow solver to solve your flow analysis more quickly.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	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
Toolbar	**Advanced Simulation** ® **Solve** ® **Edit Solution Attributes**
Menu	**Analysis** ® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the solution node ® **Edit**
Location in dialog box	**Solution Details** tab ® **Solver Selection** list ® **Parallel Solver**

NX Thermal and Flow DMP

What is it?

NX Thermal and Flow DMP is a new add-on product to the NX Thermal and Flow, NX Electronic Systems Cooling, and NX Space Systems Thermal products.

NX Thermal and Flow DMP removes any software limitations on the number of solver processes per run for parallel processing, and provides access to more computing power for thermal and flow analyses. It also enables parallel solutions over networks and clusters.

Without the new DMP license, access to up to 8 processes per run, on a single workstation, is possible if you have advanced thermal or flow licenses.

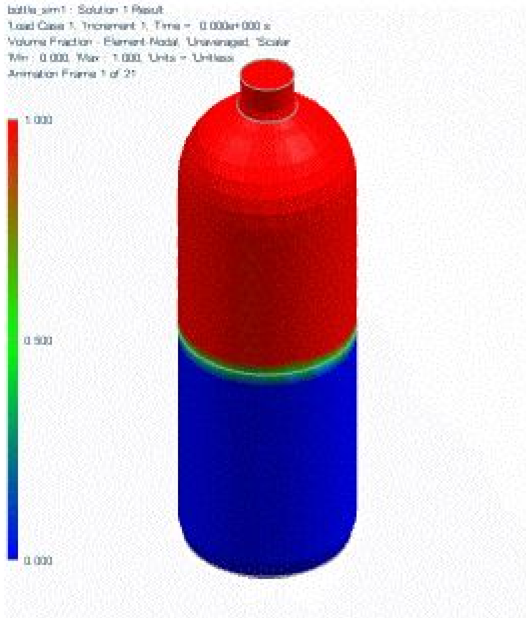
The following modules can be run in parallel:

- Flow solver
- View factor calculations

Two-phase, immiscible fluid simulation

What is it?

You can now model two-phase, immiscible fluid simulation problems in NX, such as water sloshing in a bottle or fuel levels in a gas tank.



To model sloshing, you must do the following:

1. Create two separate bodies, one for the gas region and one for the liquid region.

2. Mesh the two regions using either of the following:

   - Two 3D meshes in a single 3D mesh collector. One mesh is for the gas region and one mesh is for the liquid region. You use NX meshing tools to create the meshes.

   - A **Fluid Domain** simulation object.

3. Create a **Heterogeneous Gas-Liquid Mixture** modeling object in the 3D mesh collector or in the **Fluid Domain** simulation object.

4. Create an **Initial Gas-Liquid Mixture — 3D Flow** type of **Initial Conditions** constraint to define the liquid region.

5. Define a time-varying acceleration using a **Translating Frame of Reference** simulation object for the heterogeneous gas-liquid mixture.

For more information, see Two-phase, immiscible fluid simulation.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow

Heterogeneous Gas-Liquid Mixture modeling object

What is it?

Use the **Heterogeneous Gas-Liquid Mixture** modeling object to model a gas-liquid mixture with a clear interface between the gas and the liquid. You need to specify one gas material and one liquid material.

Both gases and liquids are materials for which the **Type** option is set to **Fluid**. The flow solver differentiates between a gas and a liquid as follows:

- When the gas constant, R_s, is defined in the **Fluid Material** dialog box, the fluid material is a gas.
- When the gas constant, R_s, is *not* defined in the **Fluid Material** dialog box, the fluid material is a liquid.

You can define a **Heterogeneous Gas-Liquid Mixture** modeling object in either of the following places:

- The 3D mesh collector which contains meshes for both the gas and liquid regions.
- The **Fluid Domain** simulation object.

For more information, see Heterogeneous Gas-Liquid Mixture.

Why should I use it?

You need to specify a **Heterogeneous Gas-Liquid Mixture** modeling object when you want to model sloshing problems.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	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
Toolbar	**Advanced Simulation** ® **Modeling Objects** ® **Heterogeneous Gas-Liquid Mixture** ® **Create**
Menu	**Insert** ® **Modeling Objects** ® **Heterogeneous Gas-Liquid Mixture** ® **Create**

Initial Gas-Liquid Mixture — 3D Flow constraint

What is it?

Use the new **Initial Gas-Liquid Mixture — 3D Flow** type of **Initial Conditions** constraint to define the liquid region in a gas-liquid mixture.

For more information, see Initial Conditions.

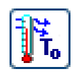
Why should I use it?

When you model sloshing, you must define an **Initial Gas-Liquid Mixture — 3D Flow** type of **Initial Conditions** constraint to differentiate the liquid region from the gas region.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	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
Toolbar	**Advanced Simulation** ® **Initial Conditions** 
Simulation Navigator	Right-click the **Constraint Set** container node ® **New Constraint** ® **Initial Conditions**
Location in dialog box	**Type** ® **Initial Gas-Liquid Mixture — 3D Flow**

Acceleration boundary condition

What is it?

Use the new **Translating Frame of Reference** type of **Moving Frame of Reference** simulation object to define an acceleration on the fluid domain. The fluid must be a heterogeneous gas-liquid mixture. The acceleration that you define is a function of time.

For more information, see Moving Frame of Reference.

Why should I use it?

Define the time-varying acceleration when you want to model sloshing problems.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow

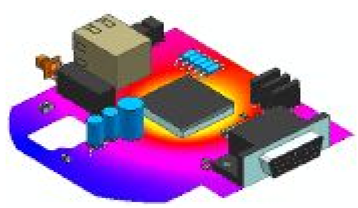
Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Moving Frame of Reference** 
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object** ® **Moving Frame of Reference**
Location in dialog box	**Type** ® **Translating Frame of Reference**

Defining printed circuit board and electronic components

What is it?

In an Electronic Systems Cooling solution, you can now more easily define the conductive and convective properties of a printed circuit board (PCB), and the thermal models of the electronic components.



- In the FEM, you model the PCB using a **Shell** type of 2D collector that points to a **PCB Stack** physical property table. The **PCB Stack** physical property table lets you define all board information, whether your PCB is a simple one-layer model or a complex multi-layer model.

- In the Simulation file, you model the PCB conductive and convective properties using the **Printed Circuit Board** simulation object and the electronic components using the **PCB Component** simulation object. The **PCB Component** simulation object lets you define thermal models and heat loads on the components, generate reports, and retrieve component thermal data from a component catalog.

Why should I use it?

These new features let you prepare a printed circuit assembly model quickly, easily, and directly inside the Advanced Simulation application. Through NX PCB Exchange, you can easily import ECAD information or update existing PCB and component data.

In addition, NX provides a sample electronic components catalog to which you can add your custom components.

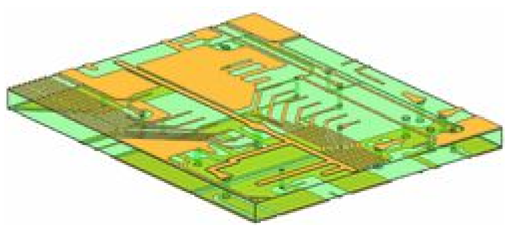
Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC

PCB Stack physical property table

What is it?

Use the new **PCB Stack** physical property table to define physical characteristics of 2D shell elements that represent a printed circuit board (PCB). The **PCB Stack** physical property table lets you define all board information, whether your PCB is a simple one-layer model or a complex multi-layer model.



In the **PCB Stack** dialog box, you can specify:

- One or more layers using the **PCB Layer** modeling object. You must define at least one layer to create the **PCB Stack** physical property table.
- One or more thermal connection between signal layers due to vias using the **PCB Via** modeling object.
- The stacking order of all PCB layers.

For more information, see **PCB Stack** physical property table.

Why should I use it?

You can model a printed circuit board's complex in-plane conductivity in each layer and across layers without using 3D solid elements.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Physical Properties** ® **PCB Stack** ® **Create**
Menu	**Insert** ® **Physical Properties** ® **PCB Stack** ® **Create**
Simulation Navigator	Right-click the 2D mesh collector node ® **Edit** ® **Type** list ® **PCB Stack** ® **PCB Stack Property** row ® **Edit** or **Create Physical**

PCB Layer modeling object

What is it?

Use the new **PCB Layer** modeling object to define the orthotropic thermal conductivity of one layer of a printed circuit board (PCB). You can define a **PCB Layer** modeling object only from a **PCB Stack** physical property table.

In the **PCB Layer** dialog box, you specify:

- The trace material for the PCB layer.
- The dielectric material for the PCB layer.
- The thickness of the PCB layer.
- The coverage of the trace material with respect to the PCB layer area. The coverage can be a value between 0 and 1 or a spatial field.

- The calculation method of the layer's orthotropic thermal conductivity. The thermal conductivity can be calculated either from the material and trace coverage or as a thermal conductivity spatial field. When you define the thermal conductivity as a spatial field, you also need to specify the material orientation.

Note You can access the **PCB Layer** modeling object only through the **PCB Stack** physical property table. You cannot access it through the **Modeling Object Manager**.

For more information, see **PCB Layer** modeling object.

Why should I use it?

A **PCB Layer** modeling object lets you easily define the variability of the PCB layer's orthotropic thermal conductivity.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Physical Properties** ® **PCB Stack** ® **Create**
Menu	**Insert** ® **Physical Properties** ® **PCB Stack** ® **Create**
Simulation Navigator	Right-click the 2D mesh collector node ® **Edit** ® **Type** list ® **PCB Stack** ® **PCB Stack Property** row ® **Edit** or **Create Physical**
Location in dialog box	**Stack Definition** tab ® **Create**

PCB Via modeling object

What is it?

Use the new **PCB Via** modeling object to define additional thermal connections between layers of a printed circuit board (PCB). You can define a **PCB Via** modeling object only from a **PCB Stack** physical property table.

In the **PCB Via** dialog box, you specify:

- The PCB layer from which the PCB via starts.
- The PCB layer to which the PCB via extends.
- The PCB via material.
- The fraction of the board area that is occupied by the vias. The fraction can be a value between 0 and 1 or a spatial field.

Note You can access the **PCB Via** modeling object only through the **PCB Stack** physical property table. You cannot access it through the **Modeling Object Manager**.

For more information, see **PCB Via** modeling object.

Why should I use it?

A **PCB Via** modeling object lets you model the thermal conductivity across layers in a PCB.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Physical Properties** ® **PCB Stack** ® **Create**
Menu	**Insert** ® **Physical Properties** ® **PCB Stack** ® **Create**
Simulation Navigator	Right-click the 2D mesh collector node ® **Edit** ® **Type** list ® **PCB Stack** ® **PCB Stack Property** row ® **Edit** or **Create Physical**
Location in dialog box	**Vias Definition** tab ® **Create**

Printed Circuit Board simulation object

What is it?

Use the new **Printed Circuit Board** simulation object to define how the heat from a printed circuit board (PCB) is transferred to the environment.

You must define a **Printed Circuit Board** simulation object on a **Surface Region** type of simulation region.

The **Printed Circuit Board** simulation object lets you define wall friction and convection properties for top and bottom surfaces. In particular, when you set **Wall Friction** to **Rough – From Mounted Components**, NX automatically takes into account all the electronic components that are mounted on the PCB.

For more information, see **Printed Circuit Board** simulation object.

Why should I use it?

When you analyze thermal response for a printed circuit assembly, you can easily define and modify convection properties for the PCB.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Printed Circuit Board** 
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object** ® **Printed Circuit Board**

Electronic component boundary condition

What is it?

The **PCB Component** simulation object lets you define the thermal models and dissipation of the electronic component, generate reports, and retrieve thermal information from a component catalog. You can include the electronic component in a **Radiation** simulation object or a **Radiation to Environment** constraint when you define the emissivity of the electronic component.

For each component, you must create a separate **PCB Component** simulation object and specify the following:

- The **Component Region** that contains 2D elements that represent the top surface of the electronic component.

- The **Mounting Region** that points to the two-dimensional simulation region on which the component is mounted. The mounting region can be the printed circuit board that is referenced by a **Printed Circuit Board** simulation object.

- The thermal model of the electronic component that you can define manually or get from your component catalog.

You can track the temperature during the solution and create a temperature report.

The following thermal models of the electronic component are available:

- The **Zero-Resistor** model connects the electronic component with the PCB using a merge thermal coupling, with the top face as the primary region and the board as the secondary region. A dissipation is applied to the top surface.

- The **One-Resistor** model connects the electronic component to the PCB using the case board resistance value that you specify. A dissipation is applied to the top surface.

- The **Two-Resistor** model connects the electronic component with the PCB using the JEDEC two-resistor model. A dissipation is applied to a lumped mass at the center of the electronic component. This lumped mass, which represents the junction, is created internally by the solver.

In all three models, you can specify the coupling resolution between the PCB and the electronic component.

For more information, see **PCB Component** simulation object.

Why should I use it?

The **PCB Component** simulation object lets you easily define all thermal properties for each electronic component in your model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **PCB Component**
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object** ® **PCB Component**

PCB Component catalog

What is it?

You can now create your own component catalog for PCB components with zero-resistor, one-resistor, and two-resistor thermal models. The sample component catalog is stored in the *component.xml* file that is located in the *[NX_installation]\NXCAE_extras\tmg\if\catalogs* folder. The *component.xml* file that you create can be located in another folder. This folder must be specified in the `MAYA_TMG_CATALOG_DIR` environment variable.

To add a new component to the catalog, you must define:

- The name of the component in the XML tag.
- All the required record XML tags with their corresponding XML tags.

  Note All values must be entered in SI units and temperature values in Celsius.

For more information, see PCB Component catalog.

Why should I use it?

You can customize your component catalog to include your company's PCB components, and you can access them in the **PCB Component** dialog box when you model thermal analysis of printed circuit assemblies.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC

Where do I find it?

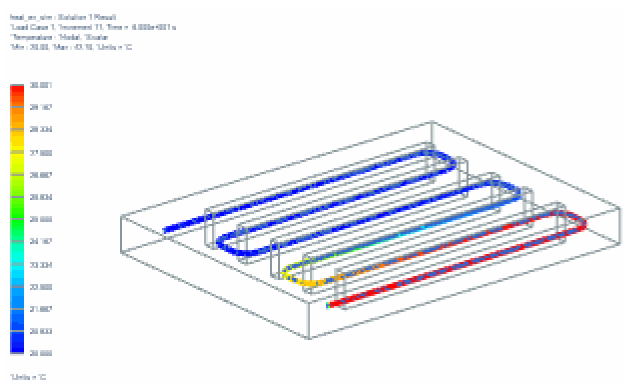
Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **PCB Component** 
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object** ® **PCB Component**
Location in dialog box	**Component Catalog** 

Duct with mass flow network

What is it?

Use the new **Duct with Mass Flow** or **Duct with Mass Flow Axisymmetric** type of 1D duct mesh collector to define fluid networks with fixed mass flows. In such fluid networks, the pressure solution that is done for normal duct elements is disabled.

As with normal duct elements, you can couple ducts with mass flow with a thermal mesh and capture the advective and convective effects.



To model a duct with mass flow network, you do the following:

1. Mesh edges, curves, and lines that represent the duct network with 1D elements.

2. Create a **Duct with Mass Flow** or **Duct with Mass Flow Axisymmetric** type of 1D duct mesh collector that contains the 1D element meshes. This mesh collector lets you define the fluid material for your network.

3. Define boundary conditions:

   - Use the **Duct Fan/Pump** type of the **Duct Flow Boundary Conditions** simulation object to define mass flow on a duct network branch.

   - Use the **Temperature** constraint or the **Duct Opening** type of the **Duct Flow Boundary Conditions** simulation object to define the temperature on desired elements or nodes in the duct network.

   - Use the **Convection Coupling** type of the **Thermal Coupling — Convection** simulation object to model the convective heat transfer from the fluid ducts to the convective regions of the thermal model.

4. Solve and post-process results.

For more information, see 1D duct flow networks.

Why should I use it?

A network of ducts with mass flow simplifies the modeling of complicated duct networks where the mass flow in branches is known and for which the pressure computation is not needed. This modeling method is useful for a number of industrial applications.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal Flow Coupled Thermal-Flow	Advanced Thermal Advanced Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Mesh Collectors** 
Simulation Navigator	Right-click the **1D Collectors** node ® **New Collector**
Location in dialog box	**Collector Type** ® **Duct** ® **Type** ® **Duct with Mass Flow** or **Duct with Mass Flow Axisymmetric**

Convection Coupling type of Thermal Coupling — Convection

What is it?

Use the new **Convection Coupling** type of the **Thermal Coupling — Convection** simulation object to model the convective heat transfer from the fluid ducts to the convective regions of the thermal model.

You specify a constant, time-dependent, or temperature-dependent heat transfer coefficient. The heat transfer coefficient can also have a spatial distribution.

You can also correct the heat transfer coefficient using the new **Duct Convection Correction** modeling object.

For more information, see **Thermal Coupling — Convection** simulation object.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation ® Thermal Coupling — Convection**
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object ® Thermal Coupling — Convection**
Location in dialog box	**Type ® Convection Coupling**

Duct Convection Correction modeling object

What is it?

Use the new **Duct Convection Correction** modeling object to correct the convective heat transfer coefficient that you specify when creating the new **Convection Coupling** type of the **Thermal Coupling — Convection** simulation object.

You can store your duct convection corrections in a correction catalog. Use the **Correction Catalog** option to add the parameters in the **Duct Convection Correction** dialog box.

A template for the correction catalog is stored in the *[NX_installation]\NXCAE_extras\tmg\if\catalogs* folder.

For more information, see **Duct Convection Correction** modeling object.

Why should I use it?

The **Duct Convection Correction** modeling object allows you to locally correct the convective heat transfer coefficient in a duct with mass flow network.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

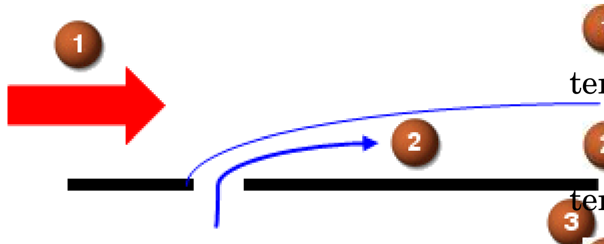
Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Modeling Objects** ® **Duct Convection Correction** ® **Create**
Menu	**Insert** ® **Modeling Objects** ® **Duct Convection Correction** ® **Create**

Film Cooling

What is it?

Use the new **Film Cooling** constraint to implicitly model the cooling of a surface by a fluid film. To create a **Film Cooling** constraint, you must specify the following:

- A convection region that is the cooled surface.
- A film reference region that contains duct elements or model edges or curves. The solver calculates the film temperature from this region.
- The convection coefficient that can vary with time and space.
- The film effectiveness coefficient that can vary with time and space.
- The free stream temperature that can vary with time.



Film cooling schematic

1 Free stream region with temperature T_{fs}

2 Fluid film region with temperature T_f

3 Convection region with temperature T_w

The film cooling constraint is similar to the convection to environment constraint, except that the adiabatic wall temperature is computed as a mixture of the free stream temperature, T_{fs}, and the fluid film temperature, T_f. The heat transfer for a convection region with a **Film Cooling** constraint is calculated as follows:

$$Q = h\,A\,(\mathrm{h}T_f + (1-\mathrm{h})T_{fs} - T_w)$$

where:

- h is the specified convection coefficient.
- A is the area of the specified convection region.
- h is the specified film effectiveness coefficient.
- T_w is the surface temperature.

For more information, see Film Cooling.

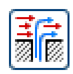
Why should I use it?

When you want to model surface temperature change due to film cooling using the **Film Cooling** constraint, you need not model all the complex interactions between the environment, the fluid film, and the geometry.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

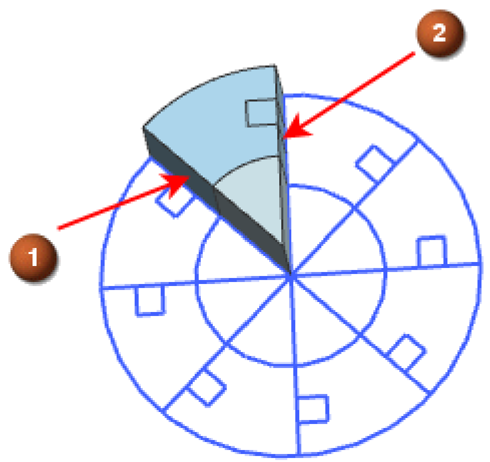
Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Film Cooling** 
Simulation Navigator	Right-click the **Constraint Set** container node ® **New Constraint** ® **Film Cooling**

Thermal cyclic symmetry using perfect contact thermal coupling

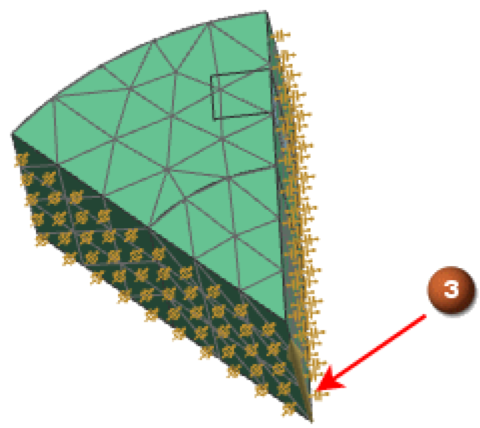
What is it?

You can now specify a rotational periodicity when you model conduction. Instead of modeling the complete geometry, you simplify it to only a small portion of the model on which you specify a **Perfect Contact** type of the **Thermal Coupling — Advanced** simulation object. You must also specify:

- The primary region and the revolved secondary region.
- The axis of revolution that is defined by a vector and a point.



Rotationally periodic portion indicating primary (1) and secondary (2) regions



Meshed solid with the Thermal Coupling — Advanced simulation object defined and showing the axis of revolution (3)

While calculating the conduction, the thermal solver revolves the secondary region (2) around the axis of revolution (3) to be in contact with the primary region (1).

For more information, see Thermal cyclic symmetry using perfect contact thermal coupling.

Why should I use it?

When you have a model that is rotationally periodic, you can reduce the number of elements by only modeling the conduction on a portion of the model. This reduces the solve time.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	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

Where do I find it?

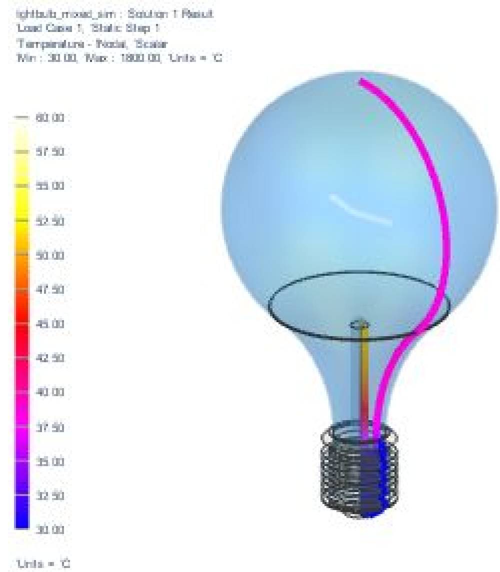
Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Thermal Coupling — Advanced** 
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object** ® **Thermal Coupling — Advanced**
Location in dialog box	**Type** ® **Perfect Contact** ® **Connection Method** list ® **Primary – Revolved Secondary**

Support of axisymmetric thermal elements in non-axisymmetric solutions

What is it?

You can now use axisymmetric elements in a thermal or coupled thermal-flow solution that also contains regular element types.

Note Only thermal elements can be axisymmetric or non-axisymmetric elements. Flow elements are always 3D elements.



Temperature distribution on an axisymmetric light bulb and a non-axisymmetric filament

Axisymmetric elements are differentiated from non-axisymmetric elements by their parent mesh collector. In the axisymmetric mesh collectors, you must specify:

- The axis of revolution
- The number of axisymmetric segments

Note Axisymmetric elements cannot be connected to non-axisymmetric elements by node sharing. You must use thermal couplings to make the connection.

For more information, see Axisymmetric modeling in non-axisymmetric solution.

Why should I use it?

When you model thermal effects on complex geometry, you can save computational time by using axisymmetric and non-axisymmetric elements together.

Supported solvers and analysis types

Solver	Analysis Type
NX Electronic Systems Cooling	Coupled Thermal-Flow
NX Space Systems Thermal	Thermal
NX Thermal and Flow	Thermal
	Coupled Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Mesh Collectors**
Simulation Navigator	Right-click the **0D Collectors**, **1D Collectors**, or **2D Collectors** node ® **New Collector**
Location in dialog box	(0D collector) **Type** ® **Lumped Mass Axisymmetric** (1D collector) **Type** ® **Beam Axisymmetric**, **Duct Axisymmetric**, or **Duct with Mass Flow Axisymmetric** (2D collector) **Type** ® **Shell Axisymmetric**

Variable thermal boundary conditions

What is it?

You can now use new independent variables when you define the magnitude of select boundary conditions.

Boundary condition	Types	Dependent variable	Independent variable					
			Time	Temp	Heat Flow Rate	Temp Diff	Thermal Capacitance	Spatial dist
Temp constraint	N.A.	Temp	Exist	N.A.	New	N.A.	New	Exist
Thermal Loads load	Heat Load	Heat flow rate	Exist	New	N.A.	N.A.	New	Exist
	Heat Flux	Heat flux	Exist	New	N.A.	N.A.	New	Exist
	Heat Generation	Heat generation	Exist	New	N.A.	N.A.	New	Exist
Joule Heating simulation object	Current	Electric current	Exist	Exist	New	N.A.	New	N.A.
	Voltage	Voltage	Exist	Exist	New	N.A.	New	N.A.
	Electrical Coupling ® Total Resistance	Electrical resistance	Exist	Exist	New	N.A.	New	N.A.
	Electrical Coupling ® Conductance Gap	Electrical resistivity	Exist	Exist	New	N.A.	New	N.A.
Override Set — Thermal Properties simulation object	Electrical Resistivity	Electrical resistivity	Exist	Exist	New	N.A.	New	Exist

Boundary condition	Types	Dependent variable	Independent variable: Time	Temp	Heat Flow Rate	Temp Diff	Thermal Capacitance	Spatial dist
Surface-to-Surface Contact simulation object	Total Conductance	Thermal conductance	New	New	New	New	New	N.A.
	Total Resistance	Thermal resistance	New	New	New	New	New	N.A.
	Heat Transfer Coefficient	Convection coefficient	New	New	New	New	New	N.A.
	Conductive Gap	Thermal conductivity	New	New	New	New	New	N.A.
Thermal Coupling simulation object	Total Conductance	Thermal conductance	Exist	Exist	New	Exist	New	Exist
	Total Resistance	Thermal resistance	Exist	Exist	New	Exist	New	Exist
	Heat Transfer Coefficient	Convection coefficient	Exist	Exist	New	Exist	New	Exist
	Edge Contact	Conductance per unit length	Exist	Exist	New	Exist	New	Exist
	Conductive Gap	Thermal conductivity	Exist	Exist	New	Exist	New	Exist
Thermal Coupling — Advanced simulation object	1 Way® Total Conductance	Thermal conductance	Exist	Exist	New	Exist	New	Exist
	1 Way ® Heat Transfer Coefficient	Convection coefficient	Exist	Exist	New	Exist	New	Exist

Why should I use it?

The new fields give you more flexibility in modeling thermal analyses.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Thermal
		Advanced Thermal
		Axisymmetric Thermal
		Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Thermal-Flow
		Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Location in dialog box	**Specify Field** list

Calculating total conductance and total resistance thermal coupling

What is it?

The way in which the thermal solver calculates the coupling value of the total conductance or the total resistance thermal coupling on the overlap region is changed. When you select the **Only Connect Overlapping Elements** check box for the **Thermal Coupling** or **Thermal Coupling — Advanced** simulation object and you specify a total conductance or total resistance value, the thermal solver now uses the total specified coupling values on the overlap area if the overlap area is not zero.

In previous releases, the thermal solver used a coupling value that was adjusted by the fraction of the overlap area over the area of the primary region.

Supported solvers and analysis types

Thermal Coupling simulation object

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling 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

Thermal Coupling — Advanced simulation object

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	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

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Thermal Coupling**  or **Thermal Coupling — Advanced** 
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object** ® **Thermal Coupling** or **Thermal Coupling — Advanced**
Location in dialog box	(**Thermal Coupling** simulation object) **Type** list ® **Total Conductance** or **Total Resistance** (**Thermal Coupling — Advanced** simulation object) **Type** group ® **1 Way** ® **Type** list ® **Total Conductance**

New thermal coupling options in Surface-to-Surface Contact

What is it?

When you use a **Surface-to-Surface Contact** simulation object to simulate a thermal contact between two face meshes, you can now use the following **Conduction Type** options to specify the conduction between the surfaces:

- **Total Conductance**
- **Total Resistance**
- **Heat Transfer Coefficient**
- **Conductive Gap**

In previous releases, you could specify only a constant heat transfer coefficient.

For more information, see **Surface-to-Surface Contact** simulation object.

Why should I use it?

The new options give you more flexibility when you model complex thermal surface-to-surface contacts.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling
		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

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Surface-to-Surface Contact** 
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object** ® **Surface-to-Surface Contact**
Location in dialog box	**Magnitude** group ® **Type** list ® **Specify** ® **Account for** list ® **Conduction** or **Conduction and Radiation** ® **Conduction Type** list

View factor calculation based on view factor residual

What is it?

You can now calculate view factors based on a target view factor residual for each element when you select the **Deterministic** calculation method for the following simulation objects:

- **Orbital Heating**
- **Radiative Heating**
- **Solar Heating**
- **Solar Heating Space**

You can also calculate view factors based on a target view factor residual for each element when you select the **Hemicube Rendering** calculation method for the **Radiation** simulation object.

Why should I use it?

This method gives you more flexibility when solving complex thermal models.

Supported solvers and analysis types

Orbital Heating and **Solar Heating Space** simulation objects

Solver	Analysis Type	Solution Type
NX Space Systems Thermal	Thermal	Space Systems Thermal

Radiation simulation object

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling 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

Radiative Heating simulation object

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	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

Solar Heating simulation object

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Orbital Heating** or **Radiation** or **Radiative Heating** or **Solar Heating** or **Solar Heating Space**
Simulation Navigator	Right-click the **Simulation Objects** node ® **New Simulation Object** ® **Orbital Heating** or **Radiation** or **Radiative Heating** or **Solar Heating** or **Solar Heating Space**

Location in dialog box	**Parameters** or **Advanced Options** group ® **Element Subdivision** list ® **Based on Error Criterion**

Catalog environment variable

What is it?

You can now store the catalogs that are used by NX Thermal and Flow, NX Electronic Systems Cooling, and NX Space Systems Thermal in any folder.

Your folder must be specified by the `MAYA_TMG_CATALOG_DIR` environment variable for NX to use it. If you set the environment variable, the catalog files must have the same names as the catalog files in the default location. The names are:

- *component.xml* which contains the PCB components catalog
- *correction.xml* which contains the duct correction catalog
- *fan.xml* which contains the fan catalog

The default location for the catalog files is the *[NX_installation]\NXCAE_extras\tmg\if\catalogs* folder.

Why should I use it?

This environment variable lets you locate the catalog files on a computer drive or network drive where you have permissions to modify files, so that you can customize the catalogs.

NX Thermal and Flow, NX Electronic Systems Cooling, and NX Space Systems Thermal customer defaults

What is it?

You can now define customer defaults for options in the **Solution** and **Solver Parameters** dialog boxes. New **Solution** and **Solver Parameters** tabs are available in the **Customer Defaults** dialog box for NX Thermal and Flow, NX Electronic Systems Cooling, and NX Space Systems Thermal.

On the **Solution** tab, you can set the defaults for the following solution options:

- Run directory and location
- Solution solve-time units
- Ambient conditions for absolute pressure, fluid temperature, and the radiative environment temperature
- Output results

On the **Solver Parameters** tab, you can set defaults for the new thermal element properties.

Why should I use it?

You can customize the thermal solver options and the solution options to follow your company standards and to best suit your analyses.

Where do I find it?

Application	Advanced Simulation
Menu	**File** ® **Utilities** ® **Customer Defaults**
Location in the dialog box	**Simulation** ® **NX Thermal / Flow** or **NX ELECTRONIC SYSTEMS COOLING** or **NX SPACE SYSTEMS THERMAL**

Mesh controls for Fluid Domain meshes

What is it?

When you mesh the fluid domain using the **Fluid Domain** simulation object, you can now better control the size, shape metric, and rate of transition between the boundary layer mesh and internal mesh structures.

- Use the **2D Element Shape Threshold** option to control the shape of the surface mesh elements. You must modify the values for the aspect ratio, the skewness, or the maximum angle.

- Use the **3D Element Shape Threshold** option to control the shape of the volume mesh elements. You must modify the values for the aspect ratio, the skewness, or the maximum angle.

- Use the **Element Shape Threshold Refinement Factor** option to control how the mesh generator refines the mesh in order to satisfy the prescribed mesh shape metrics.

- Use the **Element Growth Rate Through Volume** option to control the rate of transition of the mesh size from finer regions to coarser regions.

- Use the **Surface Curvature Based Size Variation** option to control the level of geometric approximation of the surface to generate the mesh structure.

For more information, see Fluid Domain mesh quality.

Why should I use it?

You can generate quality meshes, including boundary layer meshes, for aerodynamic applications.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Coupled Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solution** **Advanced Simulation** toolbar ® **Solve** ® **Edit Solution Attributes**
Menu	**Insert** ® **Solution** **Analysis** ® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the Simulation file node ® **New Solution** Right-click the solution node ® **Edit**
Location in dialog box	**3D Flow** tab ® **Fluid Domain Mesh Parameters** group

Global flow imbalance fraction

What is it?

Use the new **Global Flow Imbalance Fraction Option** check box to ensure that the mass and momentum imbalances are within a specified fraction. You specify the fraction in the **Global Flow Imbalance Fraction** box.

For more information, see the *Global heat and flow imbalance fractions* section of Setting flow solver parameters.

Why should I use it?

This option is an additional convergence criterion for the flow solver.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling 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
Toolbar	**Advanced Simulation** ® **Solve**  ® **Edit Solver Parameters**
Menu	**Analysis** ® **Solve** ® **Edit Solver Parameters**
Simulation Navigator	Right-click the solution node ® **Edit Solver Parameters**
Location in dialog box	**3D Flow Solver** tab ® **Convergence Control** group ® **Global Flow Imbalance Fraction Option** check box

Exponential advection method for duct thermal discretization

What is it?

A new exponential advection method is available for duct thermal discretization. This method assumes an exponential temperature profile along the length of a duct element. The exponential temperature profile is used to calculate the element average temperature and the element outlet temperature. The element outlet temperature is then used as the inlet temperature of the next downstream element in the duct network.

In previous releases, only the advection and conduction method was available. The advection and conduction method assumes a constant temperature profile along the length of the duct element equal to the center of gravity temperature. In this method, the effects of conduction and external heat are accounted for at the element inlet, where the temperature drops abruptly from the inlet temperature to the constant center of gravity temperature.

For more information, see the *Duct thermal discretization* section of Setting thermal solver parameters.

Customer Defaults

You can set the exponential advection method as the default method by setting the **Duct Thermal Discretization** option to **Exponential Advection**. This option is located in the **Customer Defaults** dialog box, under **Simulation** ® **NX Thermal / Flow** or **NX ELECTRONIC SYSTEMS COOLING** or **NX SPACE SYSTEMS THERMAL**, on the **Solver Parameters** tab.

Why should I use it?

When you solve models with pronounced variations of fluid temperature along duct networks, you can obtain more accurate temperatures without using very large number of duct elements.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal Axisymmetric Thermal Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Solve**  ® **Edit Solver Parameters**
Menu	**Analysis** ® **Solve** ® **Edit Solver Parameters**
Simulation Navigator	Right-click the solution node ® **Edit Solver Parameters**
Location in dialog box	**Thermal Solver** tab ® **Element Properties** group ® **Duct Thermal Discretization** list ® **Exponential Advection**

Element properties in Thermal Solver Parameters

What is it?

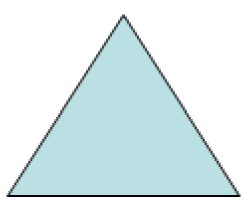
You can now access the element discretization methods in the **Element Properties** group. In previous releases, in order to access most of these methods, you had to use advanced parameters or use generic entities to modify the *INPF* file.

- The element discretization methods **Element Center Method** and **Element CG Method** are available from the **Element Discretization** list. Use these methods to calculate element conductances between beam, shell, and solid thermal elements.

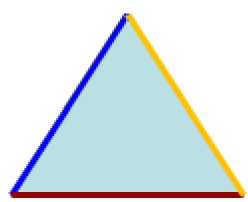
- The duct thermal discretization methods **Advection and No Conduction**, **Advection and Conduction**, and **Exponential Advection**, are available from the **Duct Thermal Discretization** list. Use these methods to calculate temperature of the duct elements. See **Title not found** for more information on the new exponential advection method.

Element Center Method	Creates conductances between element centers.
Element CG Method	(Default method in NX 8) Creates conductances between boundary elements and between the element CG and its boundary elements. When you select this method, you can also specify the methods that you want to use for **Shell Accuracy**, **Solid Element Capacitance Distribution**, and **Element Convention for Temperature Constraint**.

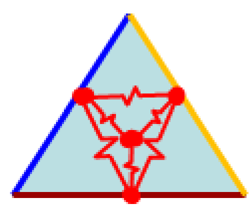
Boundary elements are created on each 3D solid element surface, each 2D planar element edge, and each 1D element end except when the surface of the solid element or the edge of a planar element is already occupied by another element. Boundary elements are only created for elements that have non-zero thermal conductivity.



Triangular element



Triangular element with its three 1D boundary elements



Conductances between the CG of the triangular element and its three 1D boundary elements

For more information, see the *Element discretization* and *Boundary elements* sections of Setting thermal solver parameters.

Customer Defaults

You can set customer defaults to specify the methods that are selected by default in the **Solver Parameters** dialog box. These options are located in the **Customer Defaults** dialog box, under **Simulation** ® **NX Thermal / Flow** or **NX ELECTRONIC SYSTEMS COOLING** or **NX SPACE SYSTEMS THERMAL**, on the **Solver Parameters** tab.

Why should I use it?

The options in the **Element Properties** group give you more control over the thermal solver to adjust parameters that suits your thermal problem.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling
		Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal
		Advanced Thermal
		Axisymmetric Thermal
		Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Thermal-Flow
		Advanced Thermal-Flow

Where do I find it?

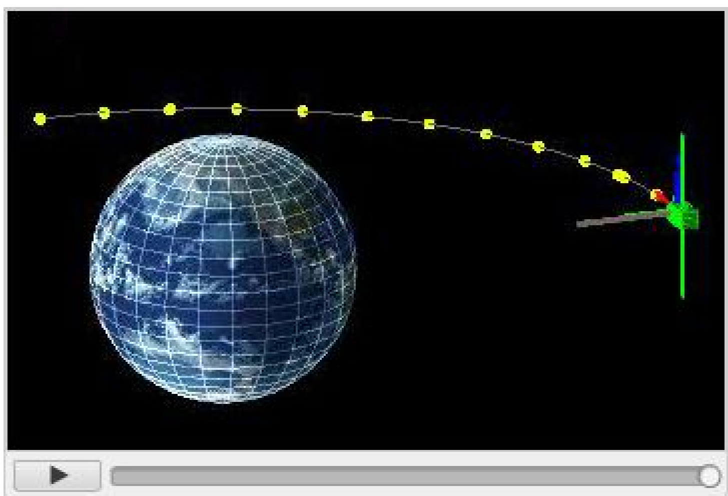
Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Solve** ® **Edit Solver Parameters**
Menu	**Analysis** ® **Solve** ® **Edit Solver Parameters**
Simulation Navigator	Right-click the solution node ® **Edit Solver Parameters**
Location in dialog box	**Thermal Solver** tab ® **Element Properties** group

Export orbit animations to videos

What is it?

You can now export videos of the orbits you animate in the Orbit Visualizer. You can manipulate the orbit display while the video is being exported. The videos can be saved in the following formats:

- ASF
- AVI
- FLV
- MOV
- MPG
- SWF
- WMV



For more information, see Using the Orbit Visualizer.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Prerequisites	A model with elements and an **Orbit** modeling object.
Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Orbital Heating**
Simulation Navigator	Right-click the **Simulation Objects Container** ® **New Simulation Object** ® **Orbital Heating**
Location in dialog box	**Display** ® **Animation** group ® **Record**

Initial condition enhancement

What is it?

When the initial conditions for the current model solution are taken from another model's solution in a different directory, the mesh and boundary conditions of the two models need not be identical. This applies both to thermal models and flow models.

For more information, see Initial conditions.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling
		Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal
		Advanced Thermal
	Flow	Flow
		Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow
		Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solution** **Advanced Simulation** toolbar ® **Solve** ® **Edit Solution Attributes**
Menu	**Insert** ® **Solution** **Analysis** ® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the Simulation file node ® **New Solution** Right-click the solution node ® **Edit**
Location in dialog box	**Initial Conditions** tab ® **Initial Conditions** list ® **From Results in Other Directory**

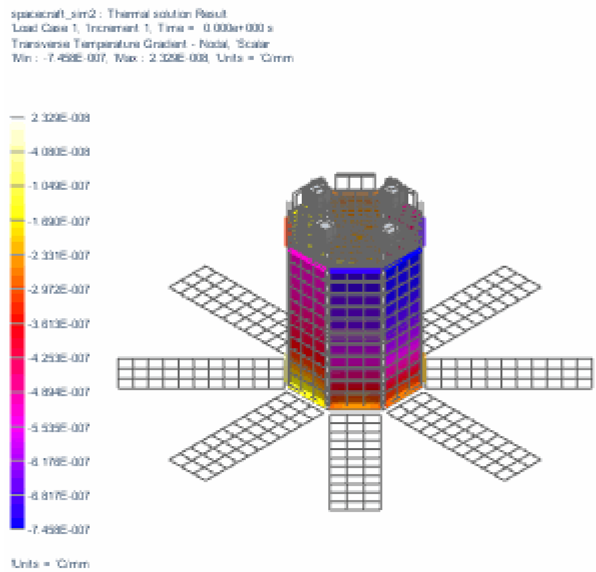
Transverse temperature gradient results set

What is it?

You can now recover transverse temperature gradient results on multi-layer shells.

After solving the thermal model, in the **Post Processing Navigator**, the following result types appear under the Solution node:

- **Transverse Temperature Gradient — Nodal**
- **Transverse Temperature Gradient — Elemental**



Transverse Temperature Gradient — Nodal result set

When you want to map the transverse temperature gradient results from a thermal solution with multi-layer shells to a structural solution, you need to select the **Transverse Temperature Gradients** check box.

For more information, see Solution dialog box — Thermal Results Options.

Why should I use it?

You can visualize the transverse temperature gradient results, which you can map to Nastran, Abaqus, or ANSYS solutions for use in structural analyses.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling 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

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solution**  **Advanced Simulation** toolbar ® **Solve**  ® **Edit Solution Attributes**
Menu	**Insert** ® **Solution** **Analysis** ® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the Simulation file node ® **New Solution** Right-click the solution node ® **Edit**
Location in dialog box	**Results Options** tab ® **Thermal** group ® **Transverse Temperature Gradients** check box

Transverse temperature gradient mapping to structural solutions

What is it?

The mapping solver can now write transverse temperature gradient results to the Nastran, Abaqus, and ANSYS input files.

- In addition to the TEMP card, the Nastran input file now contains the TEMPP1 card. For every transverse gradient element, the TEMPP1 card contains the element's temperature, which is mapped to the element's center of gravity, the elemental linear thermal gradient, and the top elemental and bottom elemental temperature.

- For every transverse gradient target node, the Abaqus input file contains the nodal temperature and the nodal linear thermal gradient.

- In addition to the BF card, the ANSYS input file now contains the BFE card. For every transverse gradient element, the BFE card contains the top and bottom elemental temperature for all N nodes on the element.

Note The transverse gradient target nodes are all the nodes selected from the **Destination Nodes** group of the **Transverse Gradient Target Set** dialog box and all the nodes connected to the elements selected from the **Destination Elements** group.

The transverse gradient target elements are all the elements selected for the **Destination Elements** group.

When you want to map the transverse temperature gradient results from a thermal solution with multi-layer shells to a structural solution, you need to select the **Transverse Temperature Gradients** check box in the **Solution** dialog box of the thermal source solution.

For more information, see Mapping results data to another model.

Why should I use it?

You can now easily map transverse temperature gradients to Nastran, Abaqus, or ANSYS solutions for use in structural analyses, such as thermal contact modeling and thermo-elastic distortion analysis.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
	Mapping	Thermal-Flow
NX Space Systems Thermal	Thermal	Space Systems Thermal
	Mapping	Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow
	Mapping	Thermal-Flow

Where do I find it?

Defining a transverse gradient pair in the source model

Application	Advanced Simulation
Prerequisite	A solution with **Analysis Type** set to **Thermal** or **Coupled Thermal-Flow**.
Toolbar	**Advanced Simulation** ® **Mapping** 
Simulation Navigator	Right-click the **Constraint Set** container node ® **New Constraint** ® **Mapping**
Location in dialog box	**Type** ® **Transverse Gradient Pair**

Defining a transverse gradient target set in the target model

Application	Advanced Simulation
Prerequisite	A solution with **Analysis Type** set to **Mapping**.
Toolbar	**Advanced Simulation** ® **Transverse Gradient Target Set** 
Simulation Navigator	Right-click the **Constraint Set** container node ® **New Constraint** ® **Transverse Gradient Target Set**

Relaxation of mapped temperature bounds

What is it?

When you map the temperatures of the source model onto the target model, you can now relax the extrapolation limits for mapped temperatures.

The maximum and minimum temperatures in the target model can now be limited by the following bounds:

$$\text{Upper limit} = T_{max} + percentage \cdot \text{abs}(T_{max} - T_{min})$$

$$\text{Lower limit} = T_{min} - percentage \cdot \text{abs}(T_{max} - T_{min})$$

where T_{max} and T_{min} are the bounds of the entire source model. The *percentage* is the relaxation of the mapped temperature limits that you specify in the **Relaxation of Mapped Temperature Limits, % (Tmax–Tmin)** box.

Why should I use it?

When there are target nodes outside the geometric bounds of the source model, you can relax the temperature limits to help extrapolate temperatures to these target nodes.

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

Where do I find it?

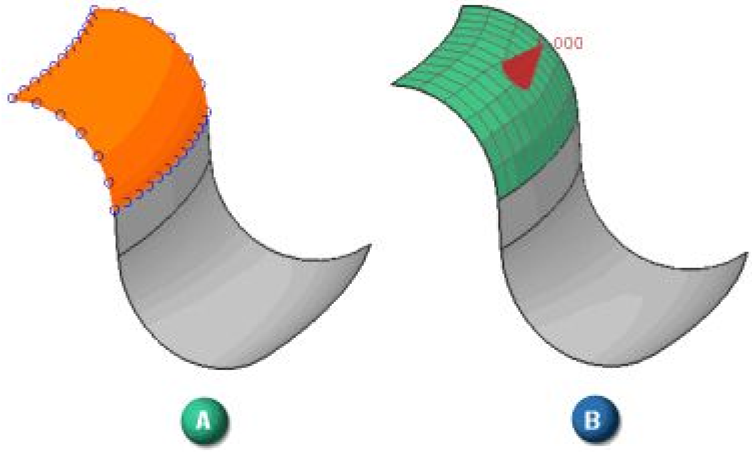
Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solution** **Advanced Simulation** toolbar ® **Solve** ® **Edit Solution Attributes**
Menu	**Insert** ® **Solution** **Analysis** ® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the Simulation file node ® **New Solution** Right-click the solution node ® **Edit**
Location in dialog box	**Mapping Details** tab ® **Settings** group

NX 8 Design Simulation

Mesh Control command now available

The **Mesh Control** command is now available from the Design Simulation application. You can use the **Mesh Control** command to more precisely control the number and distribution pattern of elements that the software creates along a selected edge or face. Edge and face densities give you local control over the number and distribution pattern of elements along a particular edge or across a particular face. You can use **Mesh Control** to create edge and face densities either before or after you have generated the mesh.

For example, you can use the **Size on Face** option in the **Mesh Control** dialog box to specify the approximate element size for the selected face. The following graphic shows the previewed nodal distribution (A) and resulting mesh (B) for a **Size on Face** density with an element size of 1mm.



Where do I find it?

Application	Design Simulation
Toolbar	**Design Simulation® Mesh Control** 
Menu	**Insert® Mesh® Mesh Control**

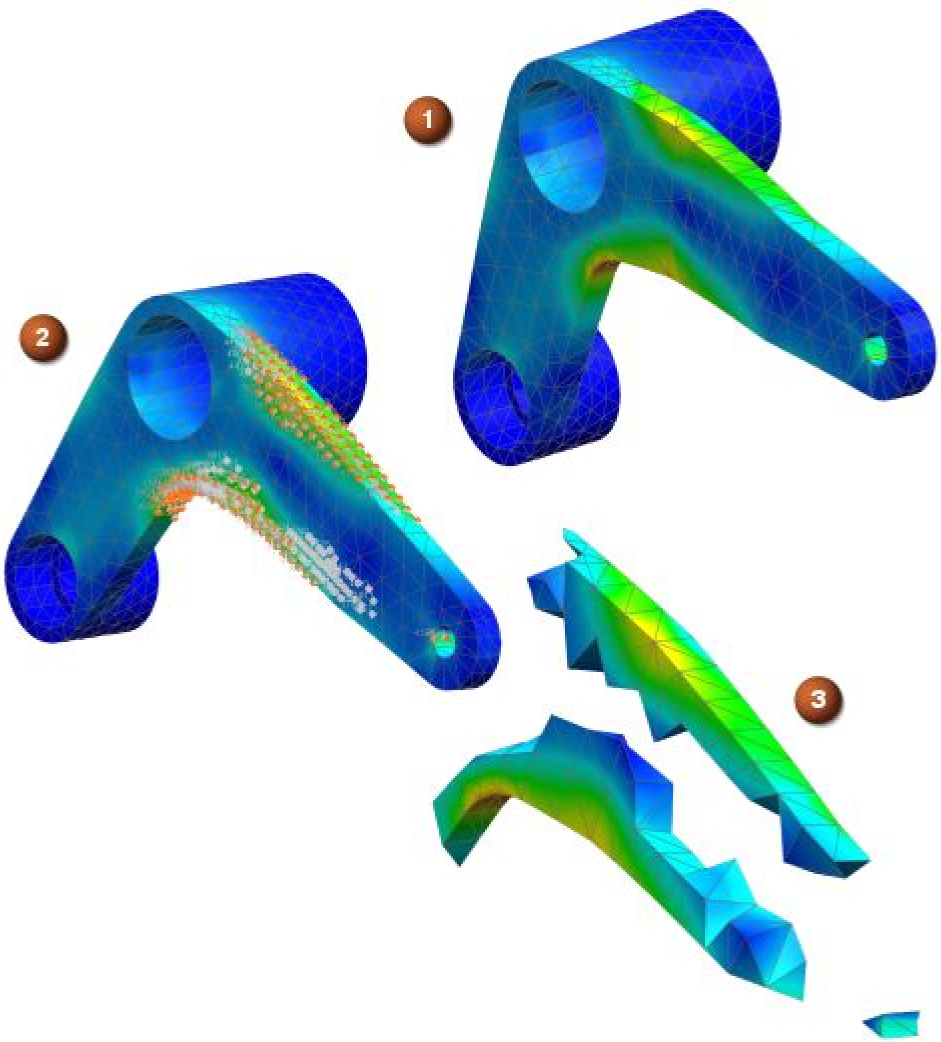
Post-processing group support

What is it?

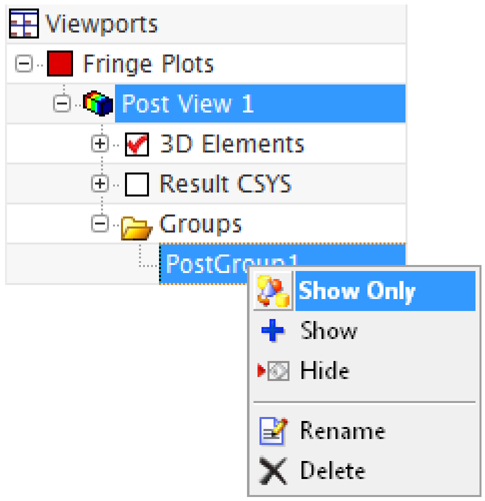
You can now use the **Identify** command to output groups for further post-processing.

You use **Identify** to probe and display nodal and elemental information for the post view display. When you have identified those nodes or elements that meet your specified criteria, click **Save selection in group**  in the **Identify** dialog box.

The software creates a temporary group named **PostGroup***n*, where *n* is an integer incremented for each group you create. Expand the post view node in the **Post Processing Navigator** to control the display of post groups.



(1) Stress results in post processing. (2) Identify nodes and elements with maximum stress, and save to a group. (3) Display the post group using Show Only.



Post groups are listed under the post view node in the Post Processing Navigator.

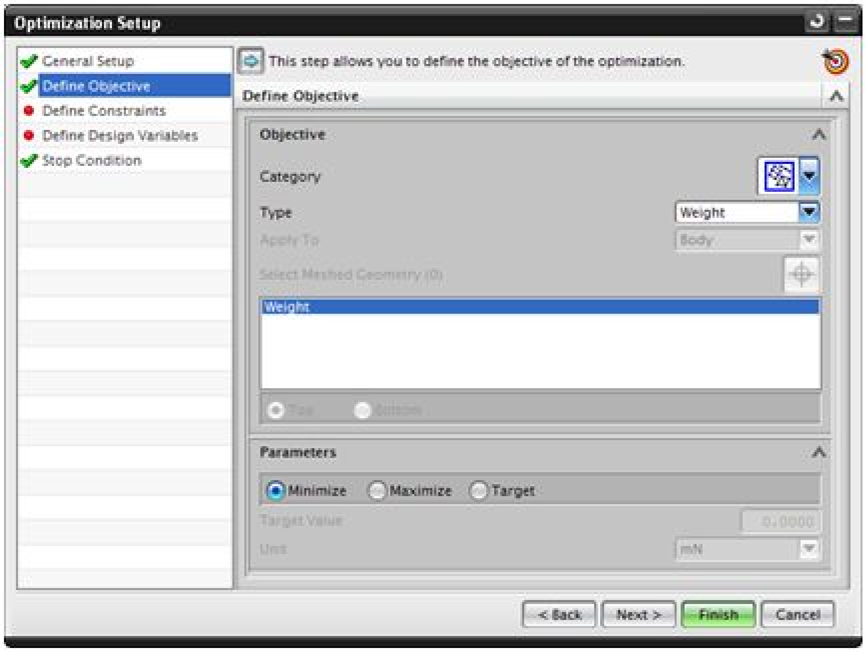
Where do I find it?

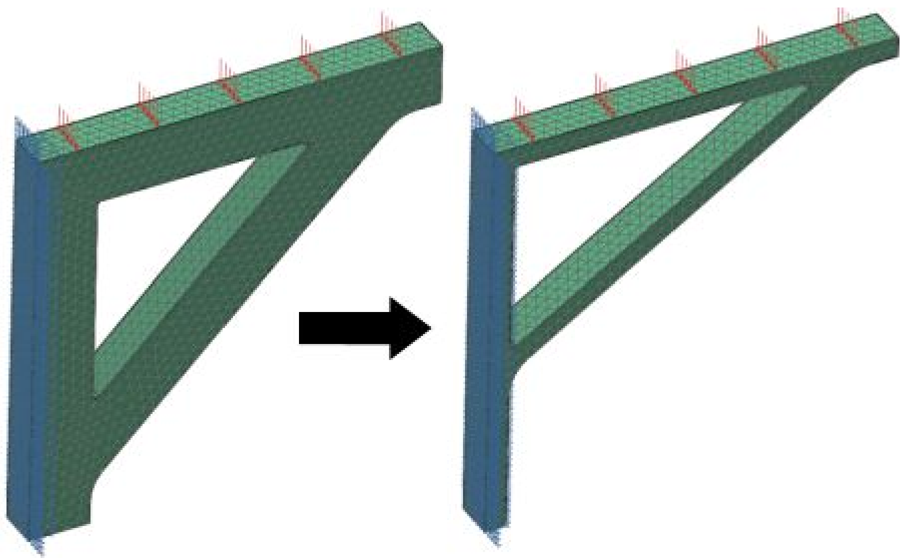
Application	Design Simulation
Prerequisite	Analysis results loaded and a post view displayed.
Toolbar	**Post-Processing** ® **Identify**
Location in dialog box	**Save selection in Group**

Geometry Optimization update

What is it?

The **Optimization** product has been rewritten in the latest NX architecture, using a wizard-like user interface that guides you through the definition of the optimization problem. The optimization algorithm has also been improved to provide a greater degree of accuracy.





Where do I find it?

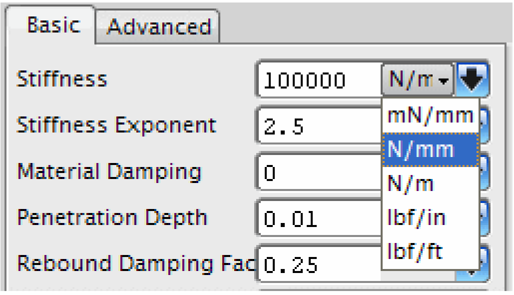
Application	Design Simulation, Advanced Simulation
Menu	**Insert® Geometry Optimization**
Simulation Navigator	Right-click the simulation® **New Solution Process® Geometry Optimization**

NX 8 Motion Simulation

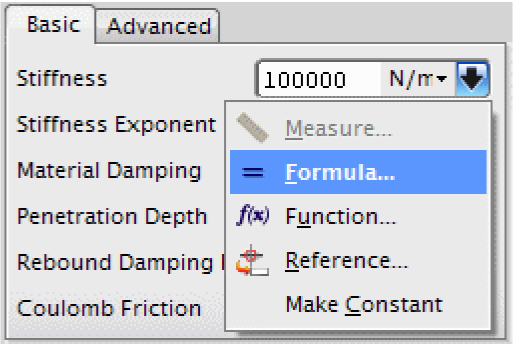
Unit and expression support

In many Motion Simulation dialog boxes, you can now change the unit of measure for many parameters. The new list options also allow you to:

- Select a recently used value for an entry field. You can edit the value that you select.
- Select a recently used formula for an entry field. You cannot edit the formula.
- Use the DesignLogic list to select a measurement, formula, or function.



Units list



Options list

Parameters in the following motion objects do not yet support this new functionality:

- Gear
- Rack and Pinion
- Cable
- 2–3 Joint Coupler

- Interference packaging
- Motor
- Signal Chart
- Solution

Checking the mechanism for errors

You can now check your motion simulation for various problems prior to solving the model, using the **Model Check** command.

The **Information** window displays messages indicating any problems with the mechanism. Explanations of typical warning and error messages are provided in the following table.

Motion object	Error message	Additional information
Link	Geometry missing	The geometry on which the link was created is no longer available. This issue may occur if the master model geometry was deleted.
	Invalid mass properties	Check the Mass Properties definition in the link.
Flexible link	Referenced .rfi/.op2 file cannot be located	The .rfi file and/or the .op2 file from the original Nastran solution associated with this flexible body solution is missing.
Joint	I/J marker is not associated with geometry	The I marker (action link) or J marker (base link) is not associative. For example, this warning may appear if you defined an orientation by specifying a fixed vector rather than inferring it from geometry. If you move the component in the master model, the joint

Motion object	Error message	Additional information
		will not be updated in the Motion simulation.
	Cannot find the function for driver	The function that was defined for the joint driver is missing.
Couplers (gear, rack and pinion, cable, 2–3 joint coupler)	More than one joint has a motion driver	Only one of the joints in the coupler definition can have a motion driver defined.
Spring, Damper, Bushing	Spring stiffness function not found	The function that was defined for the connector is missing.
	Damper coefficient function not found	
	Force stiffness/force damping/torque stiffness/torque damping function not found	
Curve on curve	Curves are not planar	The selected curves in the set are not planar.
Point on curve	No curve specified. Curve geometry lost.	The curve on which the point on curve is based is no longer available in the master model.
Point on surface	Face invalid	The software could not find the surface for the point on surface, or the surface is too complicated.
Scalar force	Function not found.	The function defined for this object is missing.
Scalar torque		
Vector object		
Signal chart		
Driver		
Marker	Marker is not smart associative.	The geometry on which the marker was created is no longer available in the master model.
Plant	Plant output function not found.	The function defined for the plant is missing.

Motion object	Error message	Additional information
Function	Function is not loaded.	
	Function is invalid.	The syntax for the function expression failed.
Solution	Loss of the .mdl file.	The software could not find the .mdl file for the solution.
	Duplicated IDs or names are found.	

Where do I find it?

Application	Motion Simulation
Toolbar	**Motion® Model Check** 
Menu	**Information® Mechanism Check® Model Check**

Exporting an animation as a movie

You can now use the **Export to Movie** command to capture your motion animation as a movie in AVI format.

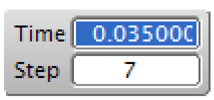
For steps, see Capture an animation as a movie.

Where do I find it?

Application	Motion Simulation
Prerequisite	A solved Normal Run, Articulation, or Spreadsheet Run solution.
Toolbar	**Motion** toolbar® **Export to Movie** 
Menu	**Tools® Animation® Export to Movie**

Animation legend

An animation legend now appears automatically when you animate the mechanism. The legend shows the current time and step during animation. The legend is interactive, so you can type in the time or step to jump to in the animation.



You can use the **Animation Legend** Motion preference to turn the legend on or off.

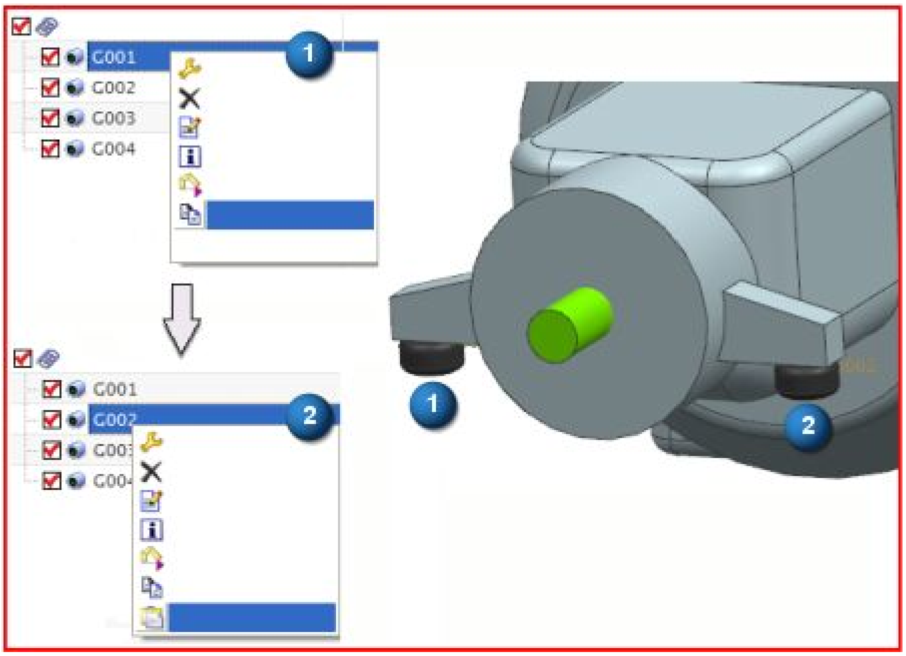
Where do I find it?

Application	Motion Simulation
Prerequisite	A solved Normal Run, Articulation, or Spreadsheet Run solution.

Copying and pasting bushing and contact parameters

You can now copy the parameters of a bushing, 2D contact, or 3D contact, and paste the parameters to another object of the same type.

1. Define the parameters of the source bushing or contact.
2. In the **Motion Navigator**, right-click the source bushing or contact and select **Copy Parameters**.
3. In the **Motion Navigator**, select the target contact or bushing and choose **Paste Parameters**.



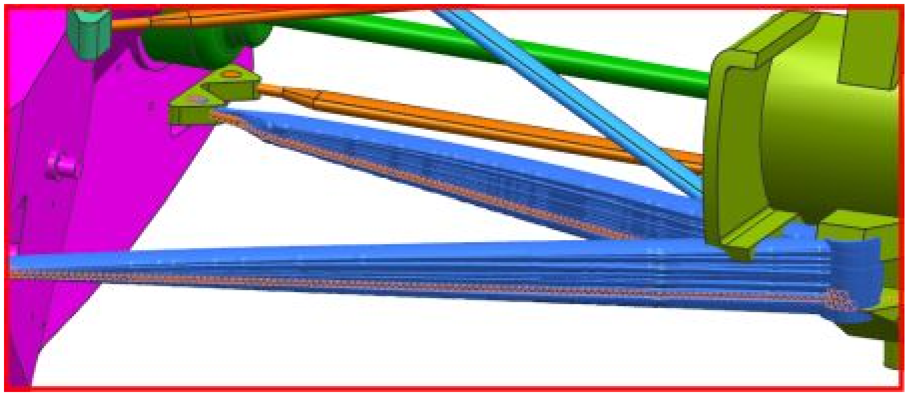
Stiffness and Damping coefficients of bushing (1) being copied to bushing (2)

Improvements to Flexible Body Analysis

Packaging options are supported

You can now define packaging options, including Interference, Measure, and Trace, in a Flexible Body analysis.

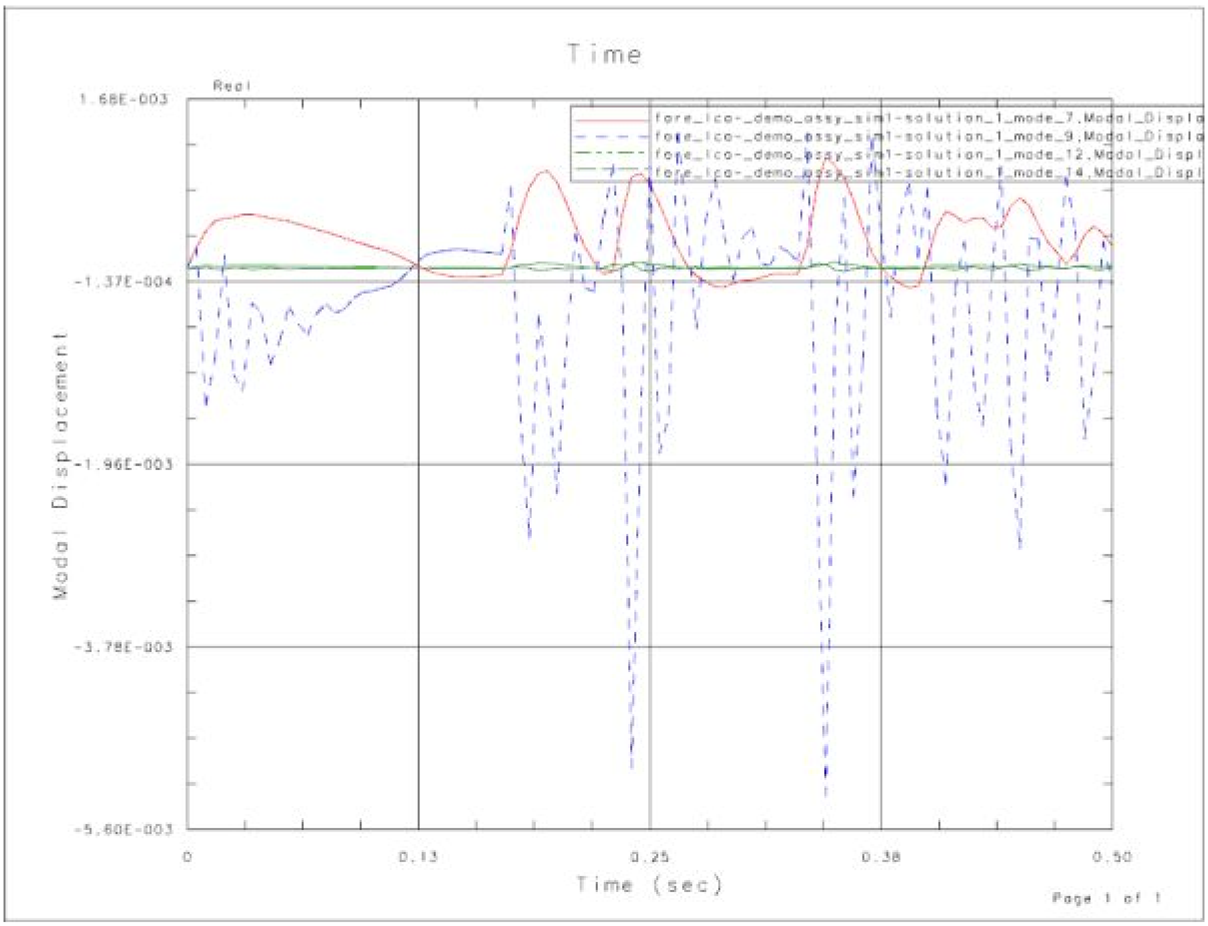
For more information about flexible body analysis, see Flexible body analysis.



Trace packaging on a flexible link

Modal coordinates can be plotted

You can now plot the modal degrees of freedom vs. time using the Modal Coordinates setting in the **Graph** dialog box. You can select one or more modes of the flexible body and then plot the modal displacement, acceleration, or velocity.



Data recovery solve can be limited to only relevant steps

You can now use the **Skip Steps** option on the **Solution** dialog box to specify steps in the solution to skip in the data recovery portion of the solve. You can greatly improve the performance of the solve by limiting the data recovery solve to only those solution steps that are relevant to the analysis.

Post View settings are persistent

During a flexible body analysis in Motion Simulation, the **Post View** settings that you define are now preserved as long as the motion simulation is open in NX. In previous releases, the default settings were restored after each animation run.

Other user interface enhancements

Easier definition of Joint and Bushing orientation

The process for specifying the orientation of a joint or bushing has been improved. You can now:

- Use the **Vector Orientation Type** to specify the orientation by selecting or defining a vector. You can select geometry to infer the vector or use the NX vector tools to specify the vector. The directions are in terms of the Work coordinate system.

- Use the **CSYS Orientation Type** to specify all three directions of the joint or bushing coordinate system using the NX coordinate system tools.

Point on Curve and Point on Surface now require link definition

When you create a Point on Curve or Point on Surface constraint, the dialog box now requires you to select the link on which to define the constraint. This link specification avoids issues with the location of the point in the constraint definition.

Curve Parameterization added to Point on Curve dialog box

The **Curve Parameterization** method has been added to the **Point on Curve** dialog box. In previous versions, this method was available only as a customer default. The customer default is no longer available.

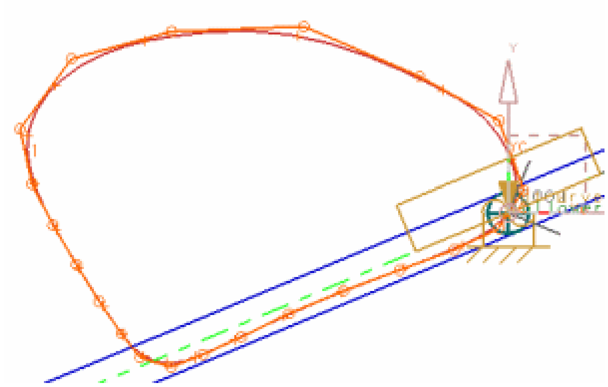
When you solve a mechanism that contains a Point on Curve constraint, NX passes the data for the curve to the solver as a group of points, using one of the following methods:

- **User-defined spacing** (Default) — Calculates the number of points based on a fixed user-defined spacing between every two points along the curve. In most cases, the default value of 2 mm is appropriate. However, if the curve has a large curvature or there is a large shift between different segments of curvature, and you have problems solving the mechanism, you may need to use a smaller value. Note that using a smaller value will slow solve performance.

  The software calculates the number of points using this formula:

  number of points = curve length / (user-defined spacing +1)

- **Curvature-based** — Calculates the number of points based on the curve itself. To check the number of points that NX uses to represent a curve, choose **Information® Object** and select the curve. The **Information** window includes the line **Number of Segments [number]** and the graphics window shows the point locations, as shown in this example.



Note In most situations, the default **User-defined spacing** option is recommended, especially if the Point on Curve constraint contains a motion driver. Use the **Curvature-based** method only if you have problems solving the mechanism and have already tried using a smaller user-defined spacing.

Improved 2D Contact and 3D Contact dialog boxes

To improve usability and simplify the definition of contact problems, the parameters on the **2D Contact** and **3D Contact** dialog boxes have been reorganized into a **Basic** tab and **Advanced** tab. For most cases, you can define the options on the **Basic** tab and use the default settings on the **Advanced** tab.

User-specified names for Motion Simulations

In the **Environment** dialog box, you can now specify a name for the motion simulation. In previous releases, motion simulations were named automatically.

NX 7.5.2 Advanced Simulation

Solver version support

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 8 releases

Solver	File Type	NX 8
NX Nastran	Import ASCII (.dat)	7.1
	Import Binary (.op2)	7.1
	Export ASCII (.dat)	7.1
	Post-processing of Results	7.1
MSC Nastran	Import ASCII (.dat)	2008r1
	Import Binary (.op2)	2008r1
	Export ASCII (.dat)	2008r1
	Post-processing of Results	2008r1
Abaqus	Import ASCII (.inp)	6.9-1
	Import Binary	N/A
	Export ASCII (.inp)	6.9
	Post-processing of Results (.fil)	6.10-1
	Post-processing of Results (.odb)	6.9-EF2
ANSYS	Import ASCII (PREP7, CDB)	12.1
	Import Binary (.rst, .rth)	12.1
	Export ASCII (.inp)	12.1
	Post-processing of Results	12.1
LS-DYNA	Import ASCII	N/A
	Import Binary	N/A
	Export ASCII (.k)	971R3.2.1
	Post-processing of Results	971R3.2.1

NX7 releases

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2
NX Nastran	Import ASCII (.dat)	6.1	7.0	7.0	7.1
	Import Binary (.op2)	6.1	7.0	7.0	7.1
	Export ASCII (.dat)	6.1	7.0	7.0	7.1
	Post-processing of Results	6.1	7.0	7.1	7.1
MSC Nastran	Import ASCII (.dat)	2008r1	2008r1	2008r1	2008r1
	Import Binary (.op2)	2008r1	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2008r1	2008r1	2008r1	2008r1
	Post-processing of Results	2008r1	2008r1	2008r1	2008r1

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2
Abaqus	Import ASCII (.inp)	6.8-1	6.9–1	6.9–1	6.9-1
	Import Binary	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9	6.9	6.9
	Post-processing of Results (.fil)	6.8-EF2	6.9.2	6.9.2	6.10-1
	Post-processing of Results (.odb)	6.8-EF2	6.9-EF1	6.9-EF2	6.9-EF2
ANSYS	Import ASCII (PREP7, CDB)	12	12.1	12.1	12.1
	Import Binary (.rst, .rth)	12	12.1	12.1	12.1
	Export ASCII (.inp)	12	12.1	12.1	12.1
	Post-processing of Results	12	12.1	12.1	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A
	Export ASCII (.k)	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

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

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
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

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
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

Selecting multiple edges or faces on an element

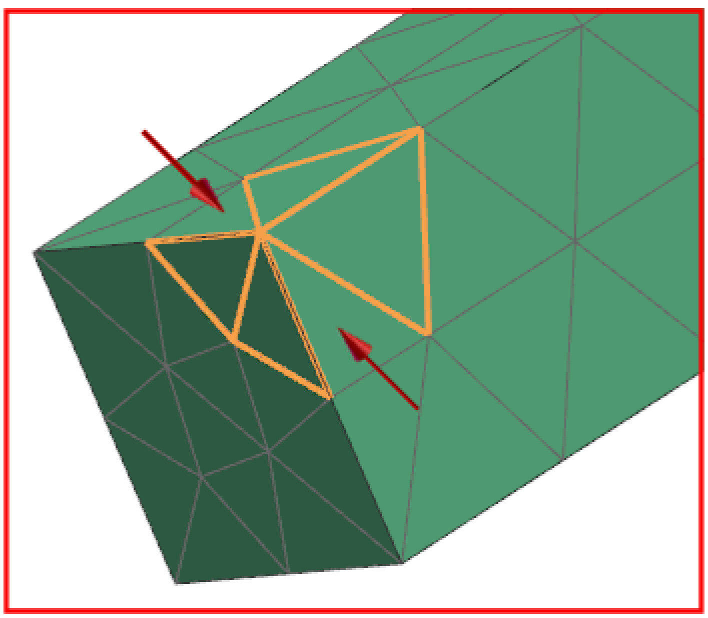
What is it?

With commands in which you can select element edges or faces, you can now select all the free faces or free edges on an element. In previous releases, you could select only one edge or face per element.

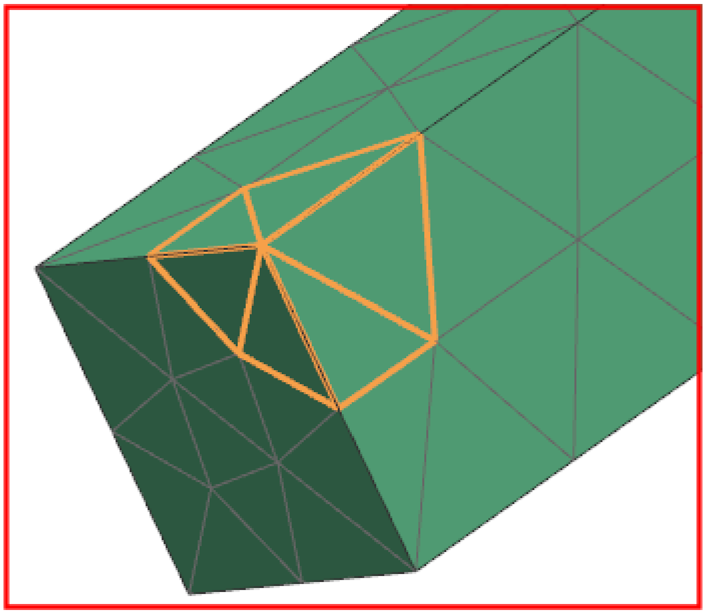
Also, in the **Type** filter (which appears in the Selection bar in the toolbar area), the **Element** filter has been renamed to **Element Face** for certain commands.

Some commands in which you can select all free edges or faces are:

- **Pressure** load
- **Region** (**Surface** type)
- **Element Associated Data**
- **Element Extrude**
- **Node/Element Information**
- **Surface Coat**



In previous releases, the element free faces indicated by arrows could not be selected.



In this release, all element free faces can be selected.

Where do I find it?

Application	Advanced Simulation Design Simulation

Simulation Navigator enhancements

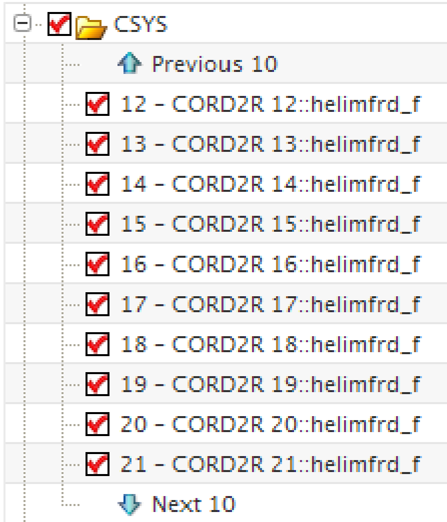
What is it?

This release includes several usability enhancements to the **Simulation Navigator**.

Paging through large numbers of objects

Using the new **Paging** option, you can now limit the number of objects that display in the navigator, such as coordinate systems, loads, and constraints. You can then page through the list of objects using the **Next** and **Previous** buttons. This option improves performance when displaying a long list (for example, 1000 coordinate systems in the **CSYS** folder).

In the **Customer Defaults** dialog box, under **Simulation® Extras® Miscellaneous**, you can control the number of options to display using the **Number of nodes per navigator page** option. The default value is **1000** objects per page.



CSYS folder with Paging turned on, with Number of nodes per navigator page set to 10

Search by entity name

Using the new **Find Object** command, you can now search for an entity by name in **Simulation Navigator** folders. When an object is found, it is selected in the **Simulation Navigator** and in the graphics window.

You can search on a partial string. For example, you can search for SHELL to find a mesh collector named PSHELL_1.

Note The search string is case sensitive. Wildcards are not supported.

Where do I find it?

Navigator Paging

Application	Advanced Simulation
Simulation Navigator	Right-click a folder® **Paging**® **On**

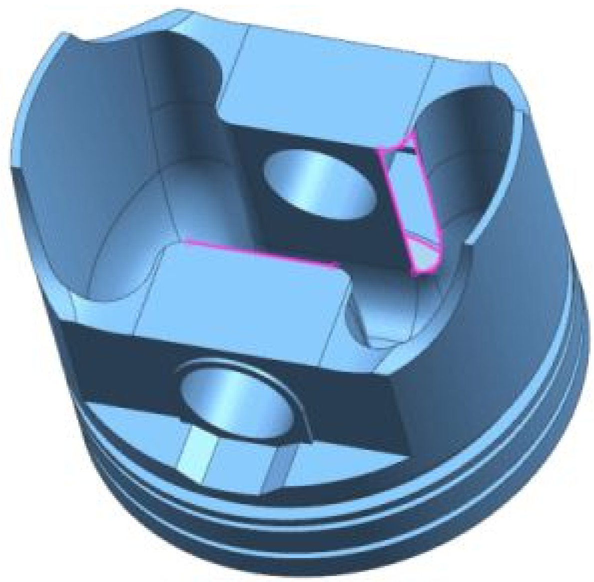
Find Object command

Application	Advanced Simulation
Simulation Navigator	Right-click a folder® **Find Object**

Displaying geometry free edges on the model

What is it?

You can now display free edges on geometry using the **Model Display** command. The new **Display Free Edges** check box is available in the **Polygon Geometry** tab, which also lets you define the color, thickness, and style of the free edge lines, as well as the marker that represents the endpoints of the edges.



Geometry free edges shown in magenta

Where do I find it?

Application	Advanced Simulation Design Simulation
Menu	**Preferences® Model Display**
Location in dialog box	**Polygon Geometry** tab

Solid properties check enhancements

What is it?

When you use the **Solid Properties Check** command to verify the properties of the elements in your model, the software now also reports:

- The total surface area of selected 2D elements in your model.
- The total length of selected 1D elements in your model.

The ability to calculate the total surface area of the 2D elements is helpful, for example, when you want to tune the mass of your model. For instance, if you have a 2D mesh on a panel, and you know that the total mass of the panel should be slightly higher, you can use the **Solid Properties Check** command to query the surface area of the 2D elements in the panel. You can then use that information to adjust the specified **Non Structural Mass** value that you specified in the **Physical Property Table** that you assigned to the mesh.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file
Menu	**Information**® **Advanced Simulation**® **Solid Properties Check**

Creating a field from an AFU function

What is it?

From the **XY Function Navigator**, you can save an XY table function in an AFU file to a table field.

After the field is created, it is listed in the **Simulation Navigator** in Advanced Simulation. You can then use this field, for example, to apply results data to a model as a boundary condition.

Where do I find it?

Application	Advanced Simulation
Prerequisite	One or more functions in an AFU file loaded or imported.
XY Function Navigator	**Associated AFU**® file name® right-click the record name® **Create Table Field**

Teamcenter Integration for Simulation

CAE migration utility

What is it?

In the traditional data model, CAE data is stored as datasets referenced by a CAD item revision. In the Teamcenter for Simulation data model, CAE data is stored as CAE-specific item revisions. In previous releases, to migrate legacy CAE data stored under the traditional data model to the Teamcenter for Simulation data model, you opened the legacy data in NX and resaved it to the new data model.

Beginning in this release, you can use the command-line utility **ugnx_caemigrate_tc** to migrate legacy CAE data to the new data model.

Using this utility, users and administrators can:

- Migrate all legacy CAE models they own.
- Specify naming conventions for migrated CAE models.

In addition, administrators can:

- Migrate all legacy CAE models across the database.
- Clean up legacy data for all users following migration.

Why should I use it?

Use the **ugnx_caemigrate_tc** command-line utility to ensure a consistent data model for all CAE data, to improve performance on opening and saving legacy CAE data in NX. Working with migrated data, you can also take advantage of the features of the Teamcenter for Simulation data model, such as improved searching and reuse.

Where do I find it?

The **ugnx_caemigrate_tc** utility in located in your *%UGII_BASE_DIR%\ugii* folder.

Material and physical properties

New dialog boxes for managing and assigning materials

What is it?

Beginning with this release, you manage your materials and assign them to parts using separate dialog boxes.

- **Assign Material** — Lets you select existing materials and assign them to selected bodies or all bodies in your part. You can also view the properties of a selected material.
- **Manage Materials** — Lets you create new materials, copy library material definitions, view and edit material properties, and load library materials into the current FEM or part file.

In previous releases, you performed these tasks using a single dialog box.

When the FEM file is displayed in Advanced Simulation, you use the **Material List** dialog box to assign a material to a physical property table, as in previous releases. The **Material List** dialog box has the same capabilities as the **Manage Materials** dialog box.

Also, beginning with this release, the **Material Library Manager** dialog box is now also available when the Simulation file is displayed in Advanced Simulation. You continue to use this dialog box to edit library material definitions, export local materials to a material library, and delete library material definitions.

Where do I find it?

Application	All
Menu	**Tools**® **Materials**® **Manage Materials** or **Assign Material**

New material types for NX Nastran solutions

What is it?

This release adds support for several new material types for use in NX Nastran solutions.

Sussman-Bathe hyperelastic material

Sussman-Bathe rubber is a nearly incompressible hyperelastic material model for NX Nastran ADVNL 601,106, ADVNL 601,129, and ADVNL 701 solutions. This model uses uniaxial stress-strain data directly instead of material constants.

For more information, see the MATHE bulk data entry (with Model field=SUSSBAT) in the *NX Nastran Quick Reference Guide*.

Shape Memory Alloy material

Shape Memory Alloy is a new material for NX Nastran ADVNL 601,106 solutions. This material can simulate the superelasticity behavior of shape memory alloy material due to the reversible phase transformation of austenite and martensite. You can use this material with the PSHELL, PSOLID, PROD, CONROD, and PCOMP physical property tables.

For more information, see the MATSMA bulk data entry in the *NX Nastran Quick Reference Guide*.

3D Orthotropic materials

You can now apply orthotropic materials to the CHEXA, CPENTA, CPYRAM, and CTETRA solid elements, for all solutions except ADVNL 601 and 701.

For more information, see the MAT11/MATT11 bulk data entries in the *NX Nastran Quick Reference Guide*.

2D Axisymmetric materials

You can now apply orthotropic materials to the CTRAX3, CTRAX6, CQUADX4, CQUADX8, and CTRIAX6 axisymmetric elements.

For more information, see the MAT3/MATT3 bulk data entries in the *NX Nastran Quick Reference Guide*.

Where do I find it?

Application	Advanced Simulation
Menu	**Tools**® **Materials**® **Manage Materials**
Location in dialog box	**New Material** group® **Type** list® select the material type (such as **Sussman-Bathe**)

New material properties

What is it?

As part of ongoing support for advanced analysis types such as nonlinear, laminates, and durability, this release adds several new material properties.

Creep properties

When Nastran, Abaqus, or ANSYS is the selected solver, the following properties for modeling creep strain are now available on the **Creep** tab in the **Materials** dialog box for isotropic, orthotropic, and anisotropic materials:

- **Kelvin-Maxwell (Rheological) Constants** — Supports Nastran creep laws **111**, **112**, **121**, **122**, **211**, **212**, **221**, and **222** as well as Abaqus creep law **HYPERB**.

- **Time Hardening (Norton-Bailey) Power Law** — Supports Nastran creep law **300**, Abaqus creep law **TIME**, and ANSYS creep law **Time Hardening**.

- **Strain Hardening Power Law** — Supports Abaqus creep law **STRAIN** and ANSYS creep law **Modified Strain Hardening**.

- **Uniaxial (Rheological) Tabular Input** — Supports input of creep as table data or definition of linear viscoelastic behavior for Nastran.

Viscoelastic properties

When Abaqus is the selected solver, you can define frequency-dependent or time-dependent properties for representing the dynamic behavior of materials such as glasses, rubbers, and high polymers under stress. For isotropic, orthotropic, anisotropic, and hyperelastic materials, Time and Frequency options are available on the new **Viscoelasticity** tab in the **Materials** dialog box.

These types of material properties let you analyze situations in which the strain rate has a significant effect on the material response.

The following domains are supported:

- **Time** — Creep Test Data, Frequency Data, Prony Series, and Relaxation Test Data definition options.

- **Frequency** — Creep Test Data, Power Formula, Prony Series, Relaxation Test Data, and Tabular definition options.

The Viscoelastic properties correspond to the parameters for the *VISCOELASTIC, *SHEAR TEST DATA, *VOLUMETRIC TEST DATA, and *COMBINED TEST DATA keywords in Abaqus. For more information, see the *Abaqus Keywords Reference Manual*.

Laminate orthotropic strength properties

In support of the Laminates capability, the following strength properties have been added for the orthotropic material type:

- Transverse shear stress limit

  S13 and S23 in Nastran

  FC,,S, XZ and FC,,S,YZ in ANSYS

- Transverse shear strain limit

  X13 and X23 in Nastran

  FC,,EPEL, XZ and FC,,EPEL,YZ in ANSYS

- Tsai-Wu interaction coefficient

  F12 in Nastran

  Failure criteria in ANSYS: This value is converted to XYCP (coupling coefficient in the FC command) by the following formula:

  XYCP = 2 * F12 * sqrt[ST1* SC1 * ST2 * SC2]

  Where ST1, ST2, SC1, and SC2 are tensile/compressive stress limits in X and Y respectively.

The new properties are located on the **Strength** tab in the **Materials** dialog box, in the **Shear Stress Limits** and **Shear Strain Limits** groups.

Advanced Durability Failure Theory properties

In the **Materials** dialog box, the **Durability and Formability** tab has been divided into two tabs: **Durability** and **Formability**. A new **Dang Van Parameters** group has been added to the **Durability** tab. You can define these properties:

- Fatigue Limit Strength in Bending
- Fatigue Limit Strength in Torsion

For more information about the Dang Van parameters, see **Unsatisfied xref title**.

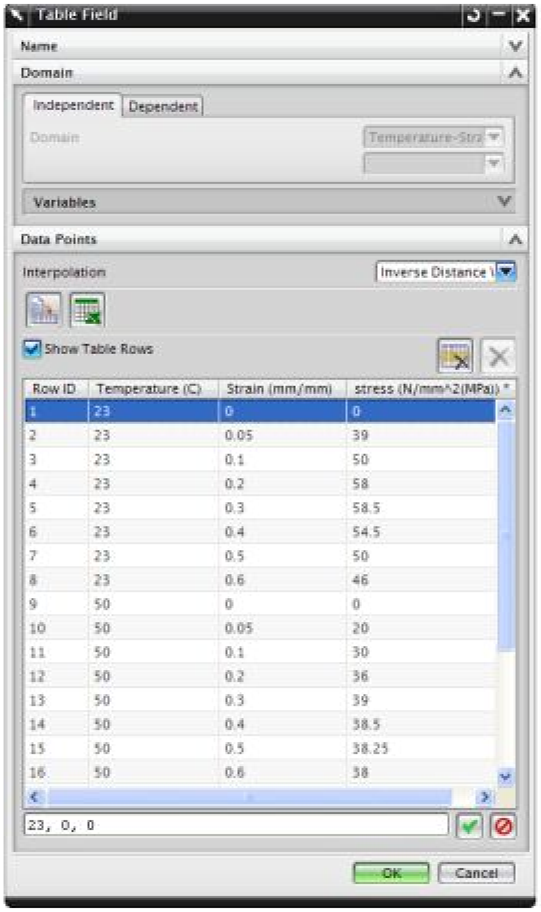
Where do I find it?

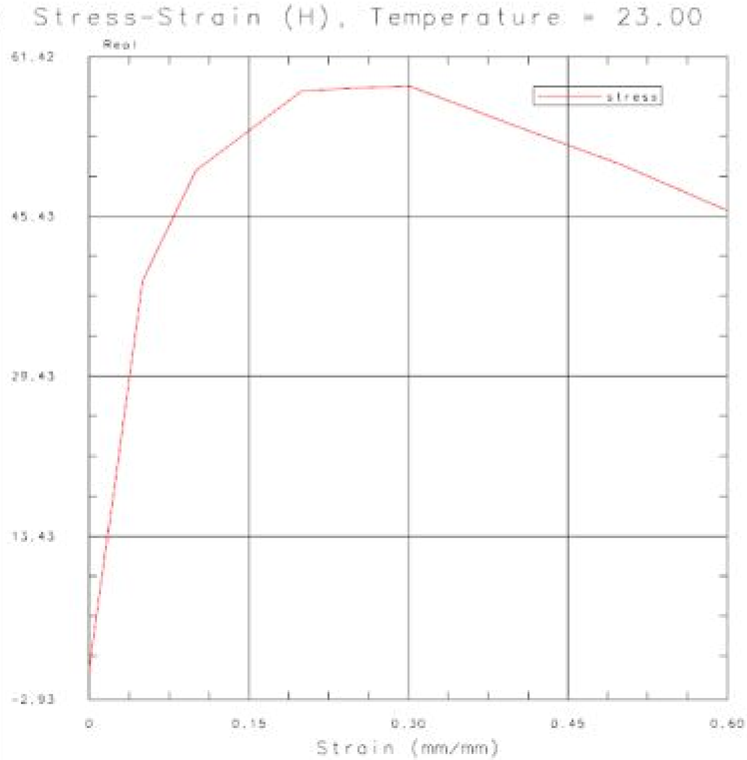
Application	Advanced Simulation
Menu	**Tools**® **Materials**® **Manage Materials**
Location in dialog box	**New Material** group® **Type** list® select the material type (such as **Sussman-Bathe**)

Temperature-dependent stress-strain curves for Isotropic materials

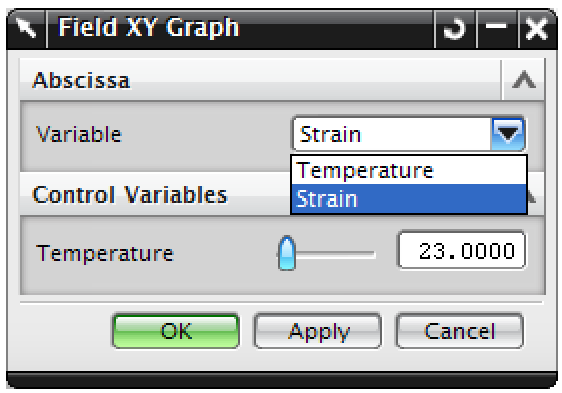
What is it?

When Nastran, Abaqus, or ANSYS is the selected solver, when you use the nonlinear elastic (NLELAST) nonlinearity type for an Isotropic material and define the tabular field data for a stress-strain curve, you can now define temperature as another independent variable. The new **Temperature-Strain** domain lets you define a table field in which each row consists of a temperature, strain, and stress value.





When you plot the field, the new **Field XY Graph** dialog box lets you specify the independent variable (either **Strain** or **Temperature**) and then specify the constant value to use in the graph.



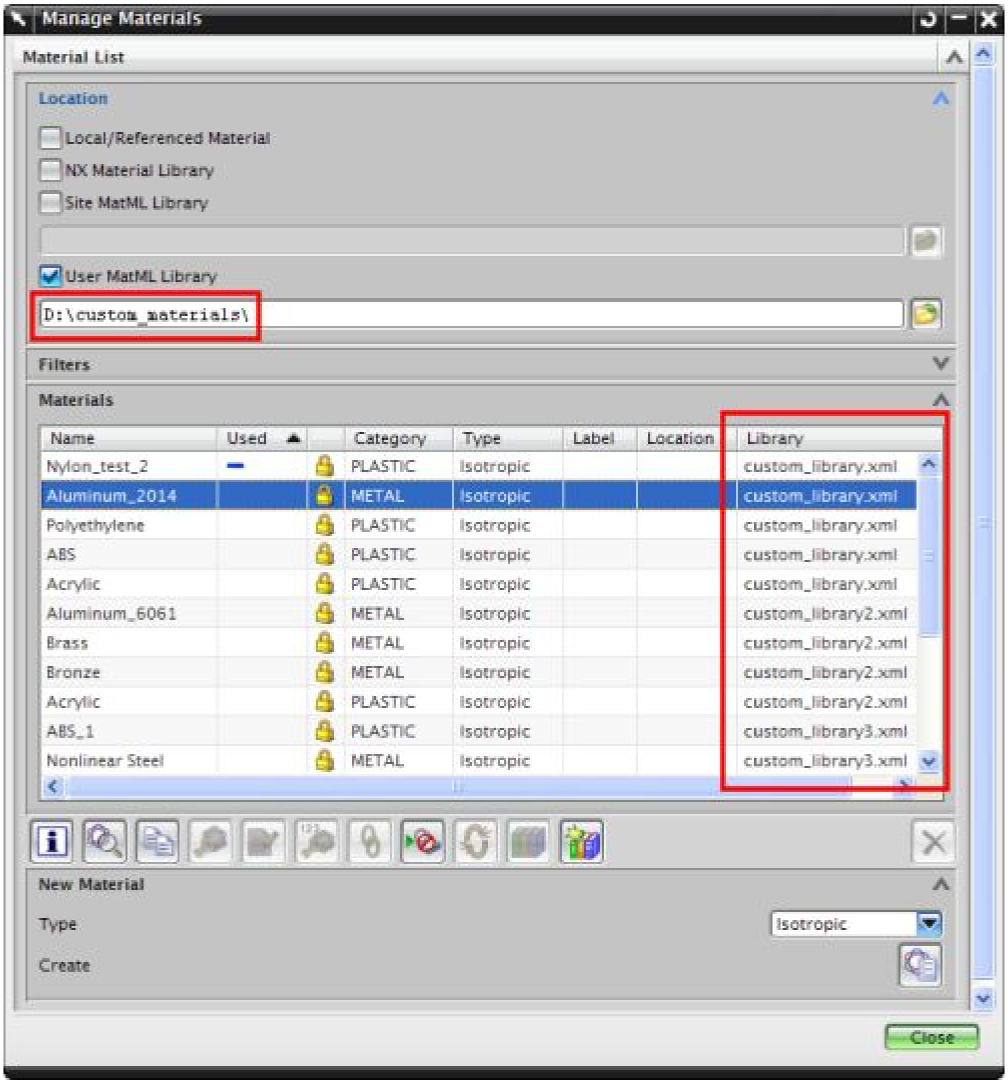
Where do I find it?

Application	Advanced Simulation
Prerequisites	Isotropic material with **Type of Nonlinearity** option set to **NLELAST**.
Menu	**Tools**® **Materials**® **Manage Materials**
Location in dialog box	**Table Field** dialog box® **Independent** tab® **Domain**=**Temperature-Strain**

Specify a directory of material library XML files

What is it?

You can now use a set of several MatML XML files as your custom material library. The new **Material Library Format** customer default controls whether the User and Site MatML libraries are single XML files or a folder containing several XML files. In the **Manage Materials** and **Assign Material** dialog boxes, the list of material definitions will include all definitions from all MatML files in the directory specified for **Site MatML Library** and/or **User MatML Library**.



Where do I find it?

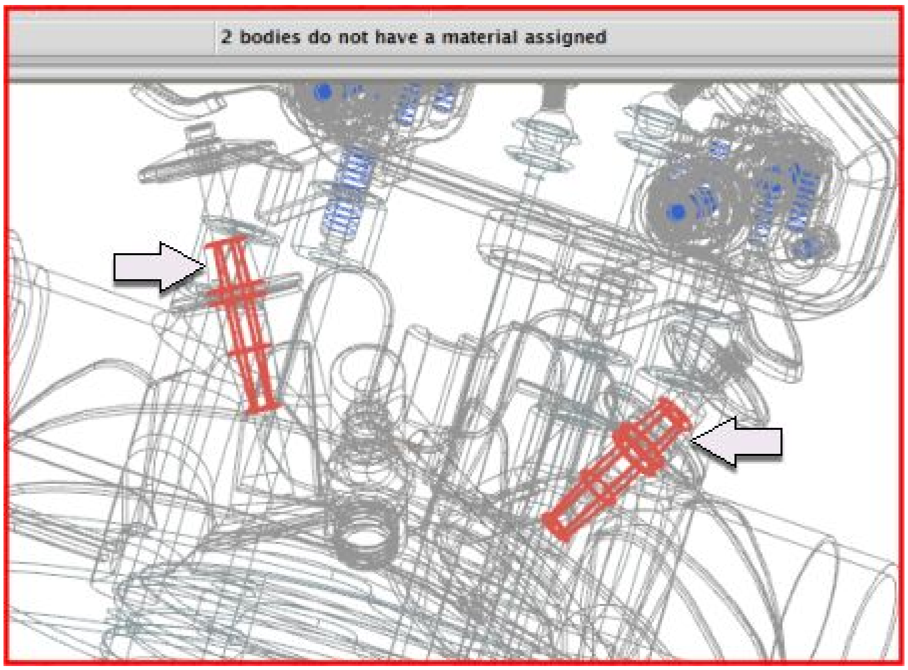
Application	All
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Customer Defaults** dialog box® **Gateway® General® Materials** tab® **Material Library Format** list = **Directory of MatML Files** You can then specify a default directory under **Site MatML Library** and/or **User MatML Library**.

Finding bodies with no material assignment

What is it?

When you manage or assign materials, you can now quickly determine which bodies in your part have no assigned material. In the **Manage Materials** and **Assign Material** dialog boxes, when you click **Highlight bodies without a material assigned** , all bodies with no assigned material are highlighted in the graphics window.

In previous releases, when a body had no material assigned, the software assigned a default density (typically that of Steel), which caused invalid mass properties in your model. This enhancement lets you ensure that the model has the correct mass properties.



Where do I find it?

Application	All
Menu	**Tools**® **Materials**® **Manage Materials** or **Assign Material**
Location in dialog box	**Manage Materials** and **Assign Material** dialog boxes® **Highlight bodies without a material assigned** 

Synchronizing loaded materials with library definitions

What is it?

You can now ensure that the material loaded into a FEM or part file always has the same properties as the library definition. In the **Material Library Manager** dialog box, when you use the **Edit Library Material** command to make changes to a library definition that is currently loaded into the FEM or part file, you can use the new **Synchronize Loaded Materials with the Modified Library Definitions** option to synchronize those editing changes back to the loaded material.

In previous releases, when you edited a library definition, the loaded material would no longer have the same properties as the library material.

Where do I find it?

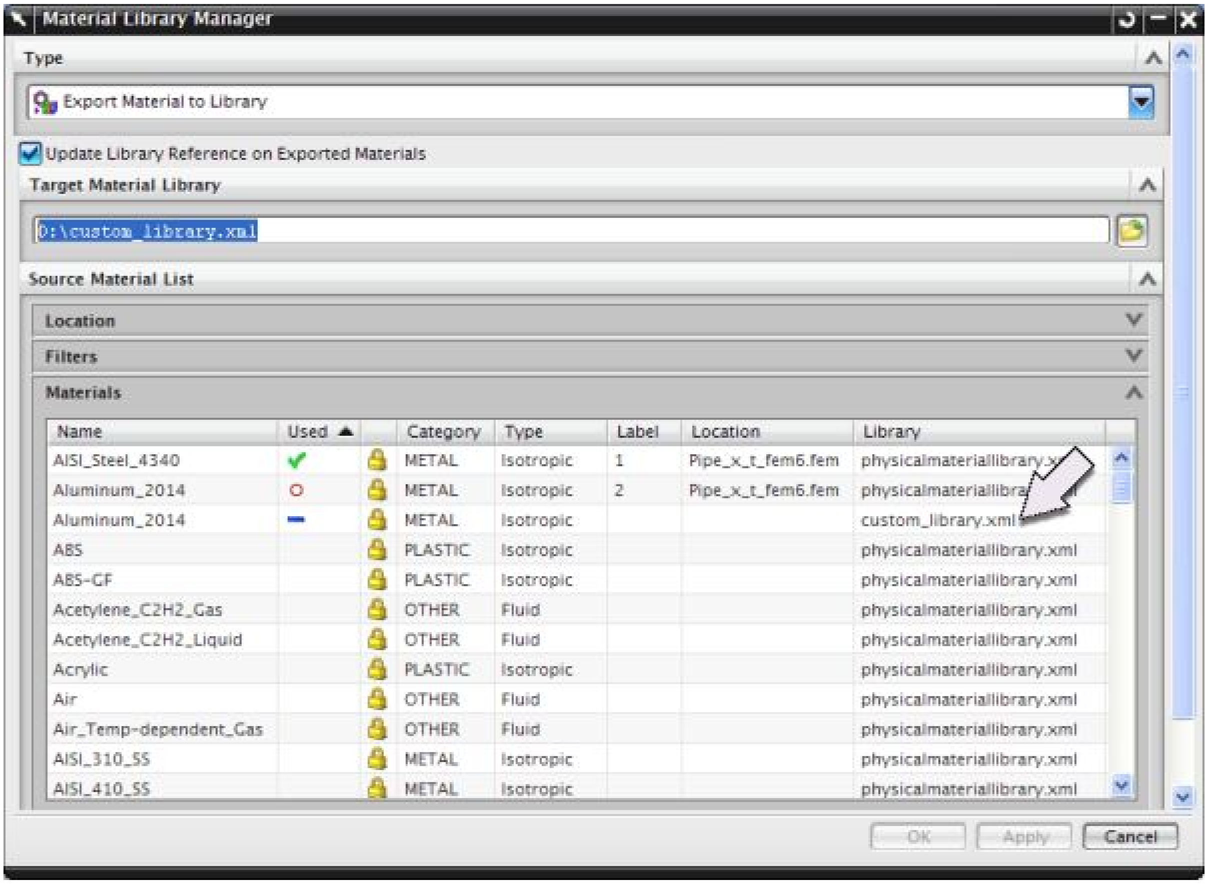
Application	All
Menu	**Tools**® **Materials**® **Manage Library Materials**
Location in dialog box	**Material Library Manager** dialog box® **Edit Library Material**® **Synchronize Loaded Materials with the Modified Library Definitions**

Library location reference for exported materials

What is it?

You can now track where your material records are stored when you use the **Export Material to Library** command to export a local material to a material library file.

Beginning with this release, you can use the new **Update Library Reference on Exported Materials** option to have the export command add the name of the target library file to the material record. The reference to the target library file appears in the **Library** column in the **Material Library Manager**, **Manage Materials**, and **Assign Material** dialog boxes.



Where do I find it?

Application	All
Menu	**Tools® Materials® Manage Library Materials**
Location in dialog box	**Material Library Manager** dialog box® **Export Library Material® Update Library Reference on Exported Materials**

Support for MatML databases from Matereality

What is it?

The NX materials functionality can now read MatML files from databases provided by Matereality®.

Where do I find it?

Application	All
Menu	**Tools® Materials® Manage Library Materials**
Location in dialog box	**Source Material List** group® **Location** subgroup® **Site MatML Library** or **User MatML Library**

Geometry abstraction

Creating a face from an existing mesh

What is it?

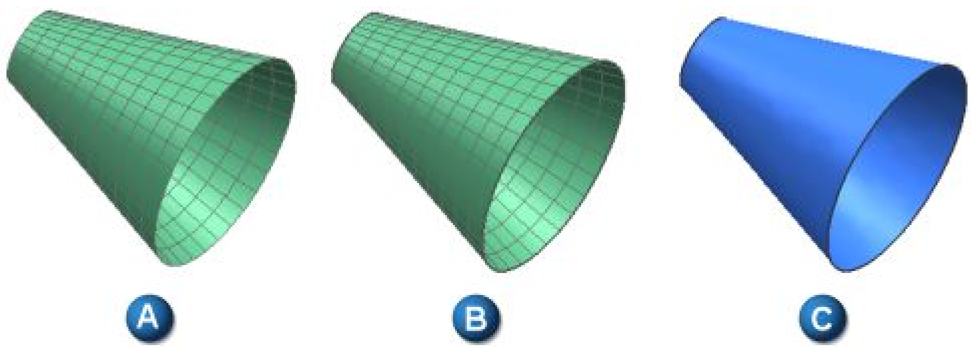
In some analyses, you may need to modify finite element models that either have no geometry or which are not associated to geometry. In previous releases, making changes to models that did not have associated geometry was challenging. You could use the tools on the **Node Operations** and **Element Operations** toolbars to make limited modifications, but those changes would not persist through a part update.

Beginning in this release, you can use the new **Face from Mesh** command to create a new polygon face from a selected set of 2D elements or 3D element faces. The elements you select to create the new face become the initial mesh on that face. You can then make modifications to the new face or its associated mesh.

The **Face from Mesh** command is helpful when you need to:

- Associate an existing finite element model to new geometry
- Connect shell meshes that are not associated with any geometry.
- Locally refine a region of a mesh that is not associated with any geometry.

The following graphic shows a very simple use of the **Face from Mesh** command. (A) shows a FEM file that was created from Nastran bulk data. The model contains only 2D PSHELL elements. (B) shows the face that was created from those elements. Those elements become the initial mesh on that new face. (C) shows the new face (with the mesh display turned off).



Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file active
Toolbar	**Advanced Simulation® Face from Mesh**
Menu	**Model Cleanup® Face from Mesh**

Meshing

Reusing a beam cross section in other parts

What is it?

You can now define a standard, **General Geometry**, or **Face of Solid** type of beam cross section and save it as a template in the NX Reuse Library so that it can be used in other FEM files for creating new beam cross sections.

For the **General Geometry** and **Face of Solid** types of cross sections, the template includes the cross sections and their user-defined stress recovery points. Values are converted to the units of the FEM in which the cross section is reused, and all association with the original sketch or solid face is removed.

When you reuse a **Face of Solid** type of cross section, the type is changed to **General Geometry**.

Where do I find it?

Saving a cross section as a reusable object:

Application	Advanced Simulation
Prerequisite	Cross section type must be standard, **General Geometry**, or **Face of Solid**.
Toolbar	**Reuse Library® Define Reusable Object**
Menu	**Tools® Reuse Library® Define Reusable Object**
Reuse Library navigator	Right-click library node® **Define Reusable Object**

Using a cross section reuse object in another FEM file:

Application	Advanced Simulation
Reuse Library navigator	**Member Select** group® click **Beam Section Template** **Search** group® enter name of reuse object and click **Search in tree above**

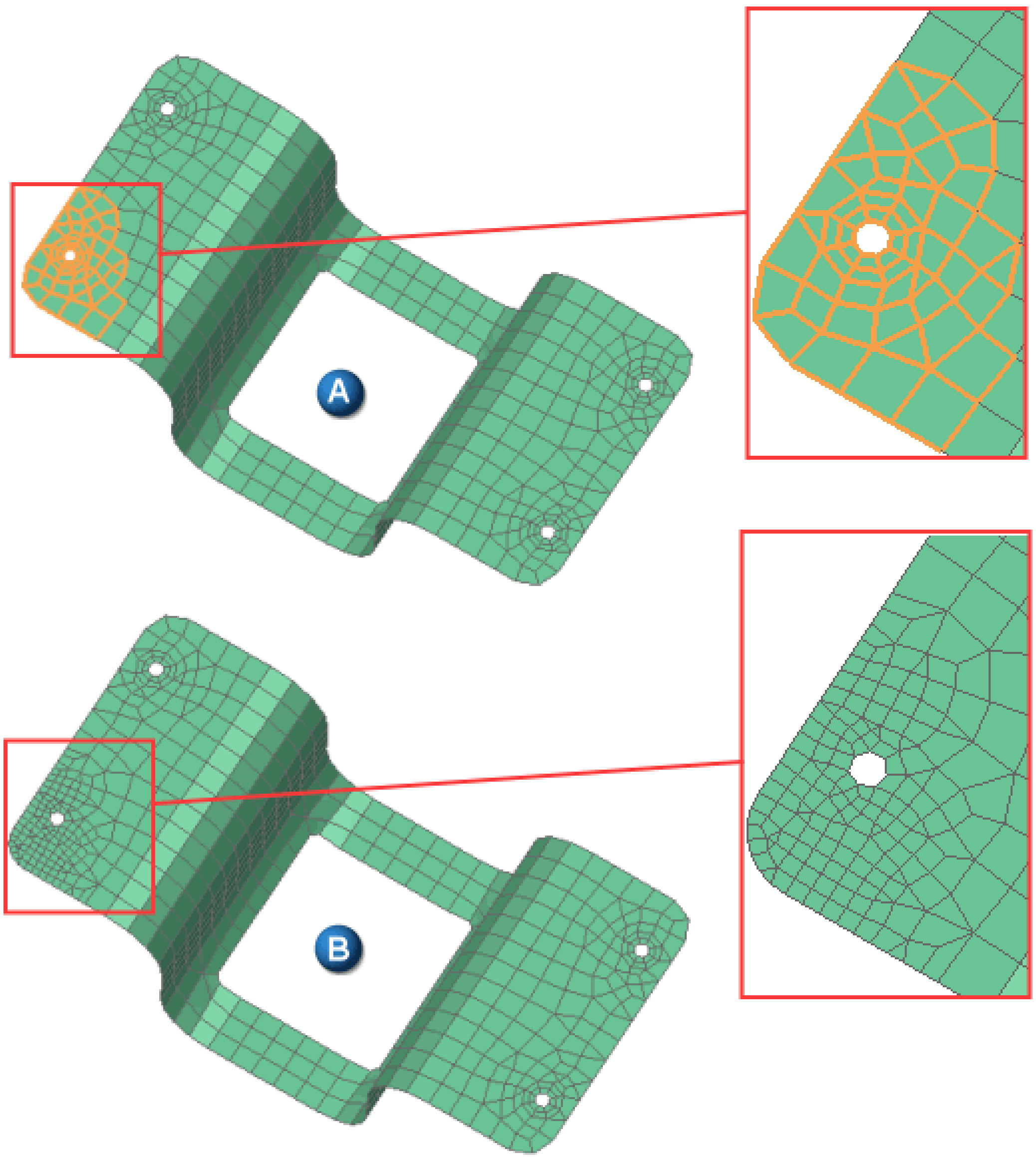
Local remeshing for 2D meshes

What is it?

For 2D meshes that are associated with geometry, you can use the new **2D Local Remesh** command to refine the elements in a very specific region without regenerating the entire mesh. For example, you may discover after you solve your model that the applied loads and structural response require that you refine the elements in a region within a face. You can then use **2D Local Remesh** to reduce the size of the elements in that region.

With **2D Local Remesh**, you select the elements to refine. You then use the options in the **2D Local Remesh** dialog box to control how the software refines the mesh. For example, you can specify a new element size or you can have the software reduce the size of the selected elements by a specified factor.

The graphic below shows a 2D mesh on the midsurface of a bracket. (A) shows the elements around one bolt hole that we selected for refinement using the **2D Local Remesh** command. (B) shows the refined mesh. Here, the selected elements were refined by a specified **Element Scale Factor** of 0.3 of their original size.

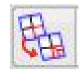


New transition method for 2D local remeshing

The **2D Local Remesh** command uses a new transition algorithm to control the rate of transition between the size of the elements in the region you refine and the size of the elements adjacent to that region. This new transition algorithm is more robust and predictable than the transition algorithm currently used by the **2D Mesh** command. You can use the **Transition** option in the **2D Local Remesh** dialog box to control the size gradient.

Note To compare the differences between the transition algorithm used by the **2D Mesh** command and the algorithm used by the **2D Local Remesh** command, you can use **2D Local Remesh** to remesh an entire existing mesh with the **Transition** option set to 1.

Where do I find it?

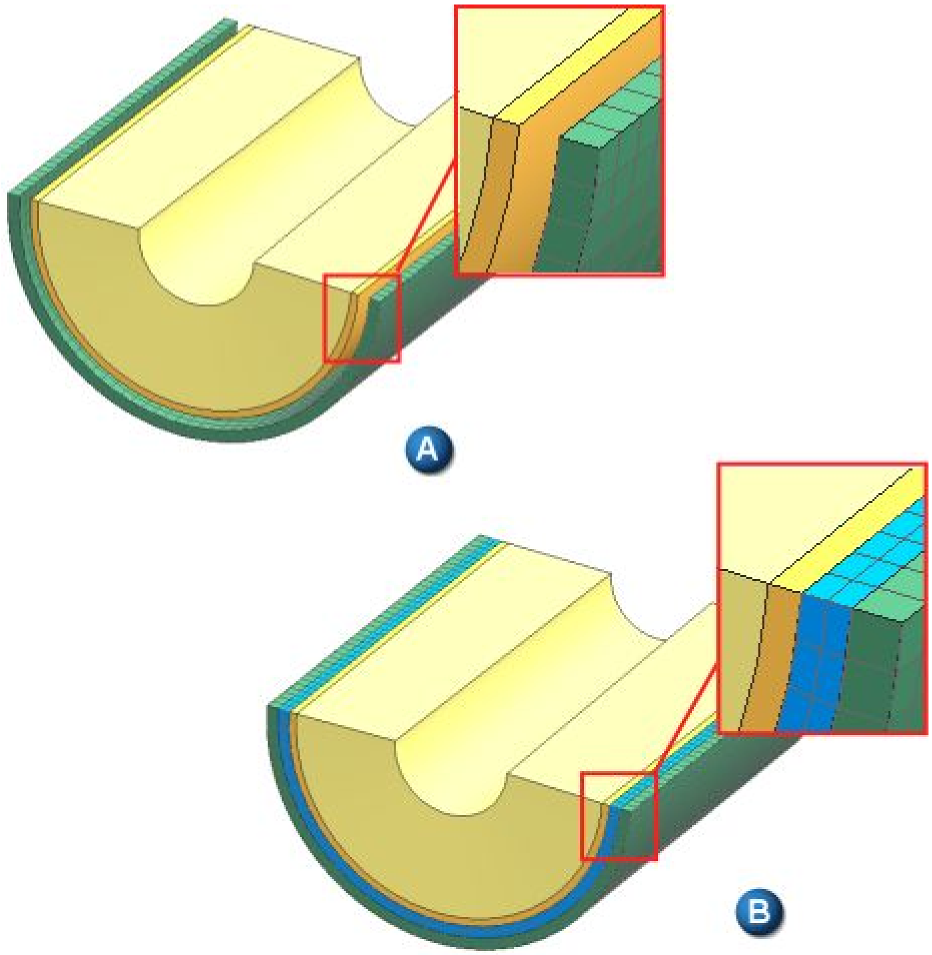
Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 2D Local Remesh** 
Menu	**Insert® Mesh® Local Remesh**

Sweeping a 3D mesh between bodies

What is it?

Use the new **3D Sweep Between** command to sweep a structured 3D mesh between two solid bodies. For example, you can use **3D Sweep Between** to create a mesh of brick or wedge elements even if you do not have any geometry to define a body. With the **3D Sweep Between** command, the software sweeps a mesh of elements from a source face to a target face. For example, if you have a set of assembly components that you need to connect but you do not have any geometry to represent the connection, you can sweep a mesh of hexahedral elements between the components.

In the following graphic, (A) shows a pressure vessel that contains a void between the outer housing and the inner volumes. Because there is no geometry in the void space, we used the **3D Sweep Between** command to manually sweep a mesh from the interior face of the outer housing to the inner volumes. (B) shows the two layers of hexahedral elements that the software generated in that void.



3D Sweep Between generates a non-associative mesh

The mesh that you create with the **3D Sweep Between** command is not associated to any geometry. The software does not update non-associative meshes if the adjacent geometry changes or remeshes. Additionally, you cannot modify a mesh that you create with the **3D Sweep Between** command. If you need to make changes to an existing manual swept mesh, you must delete the mesh and recreate it.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® 3D Sweep Between**
Menu	**Insert® Mesh® 3D Sweep Between**

Support for NX Nastran spot weld connections

What is it?

Use the new **CFAST/CWELD Connection** command to create NX Nastran CFAST and CWELD type connections in Advanced Simulation. In NX Nastran, you use CFAST and CWELD elements to model connections, such as fasteners, spot welds, or rivets, between multiple sheet bodies. You can use CFAST and CWELD elements to connect elements either within a single FE model or between FE models in an assembly FEM.

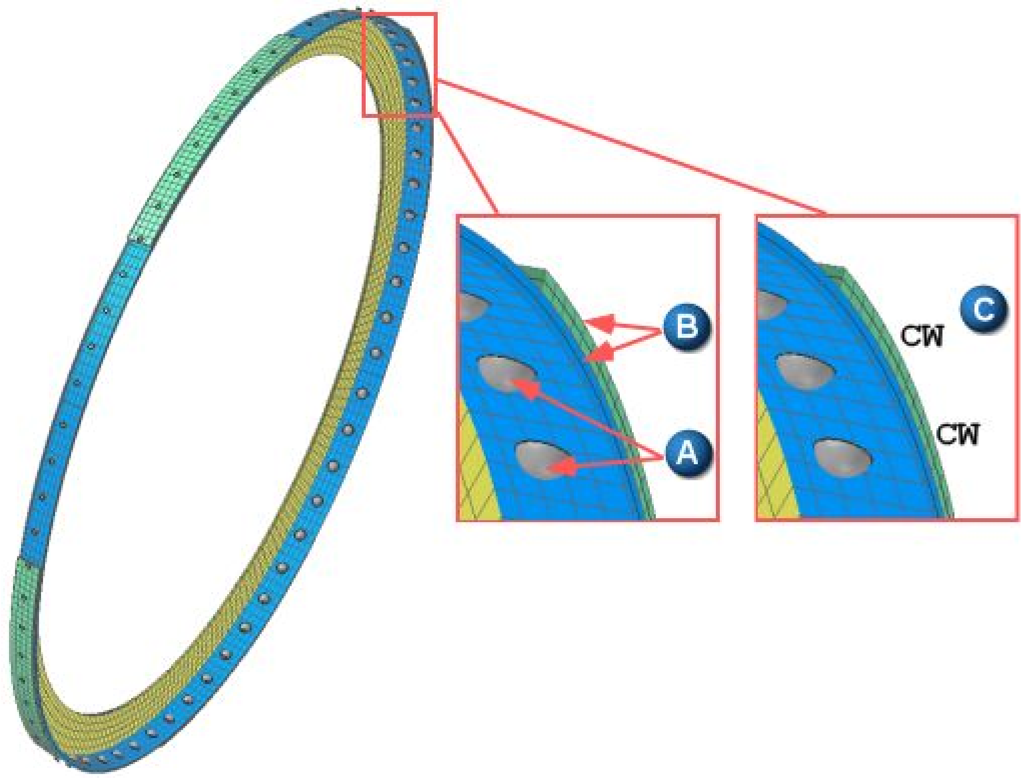
CFAST and CWELD connections:

- Are not actual elements. They are simply connection definitions. NX Nastran internally generates the constraint equations that define the stiffness for the CFAST or CWELD connections when you solve the model.

- Provide alternatives to traditional NX Nastran connection methods, such as CBAR, RBE2, or RBE3 elements or MPC constraints. In general, CWELD and CFAST connections are easier to create and use.

For more information on connection elements in NX Nastran, see *Special Element Types* in the *NX Nastran Element Library Reference Manual*.

In the following graphic, (A) shows the location of the rivets which represent the points on which to create the CWELD elements. (B) shows the two different meshes (physical property table IDs) that are connected by the CWELD elements. (C) shows the CWELD connections that the software creates at those locations.



Defining a CFAST or CWELD connection

In the new **CFAST/CWELD Connection** dialog box, you select the point or points at which you want to create the connection. Typically, these points represent the locations of spot weld or rivets. You then specify a search distance from that point. When you click **OK** or **Apply** in the dialog box, the software finds the meshes to connect by searching for unique physical property table IDs that lie within the specified distance.

In NX Nastran, you use the PFAST and PWELD bulk data entries to control the properties of the weld or fastener, including the material and diameter of the fastener or weld. In NX, you specify the properties in the new **PFAST** or **PWELD** physical property table dialog boxes.

CFAST and CWELD connections in the Simulation Navigator

The software places all CWELD and CFAST connections in the **0D Connections** node in the **Simulation Navigator**. For example:

You cannot edit an existing CFAST or CWELD connection. If you need to modify an existing CFAST or CWELD connection, you must delete the connection and recreate it.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with NX Nastran as the specified solver
Toolbar	**Advanced Simulation® CFAST/CWELD Connection**
Menu	**Insert® Mesh® CFAST/CWELD Connection**

Manual element creation enhancements

What is it?

You can now use the **Element Create** command to manually create the following additional element types:

- Linear and parabolic tetrahedrons (Tet4 and Tet 8 elements)
- Linear and parabolic hexahedrons (Hex8 and Hex20 elements)
- Parabolic pentahedrons (Wedge15 elements)

When you create these new element types, you select the nodes and, in the case of parabolic elements, the midnodes that define the element's topology. Messages in the cue line help you select the nodes in the appropriate order.

In previous releases, you could only create these types of elements using one of the geometry-based, automatic meshing commands, such as **2D Mesh** or **3D Swept Mesh**. The ability to create these types of elements manually can be helpful in cases in which your model either does not contain any CAD geometry or contains regions without any CAD geometry.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® Element Create**
Menu	**Insert® Element® Create**

Element Modify Connectivity enhancements

What is it?

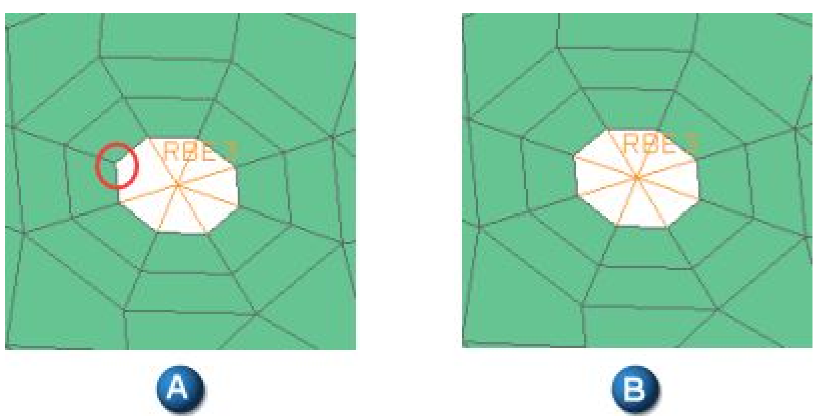
You can use new options in the **Element Modify Connectivity** dialog box to:

- Modify the leg nodes of a spider-type connection, such as a Nastran RBE3 element. For example, you can add additional leg nodes or remove existing leg nodes.
- Swap the end nodes of any two-noded 1D element.

In the new **Elements to Modify** list:

- Select **Single Element** to modify the connectivity of a single element. For example:
  - o For 2D or 3D elements, you can replace a node with another node that is not already associated with that element.
  - o For two-noded 1D elements, you can either replace a node or swap the two end nodes.
  - o For 1D connection elements, such as Nastran RBE2 or RBE3 elements, you can either replace a node or modify the leg nodes.
- Select **Elements Attached to Selected Node** to modify the connectivity of all elements attached to a single node by replacing that node with another node. In previous releases, this was the only available option.

The following graphic shows an example of how you can use **Element Modify Connectivity** to add additional leg nodes to an RBE3 element. In (A), the node at the center of the RBE3 is not connected to one of the nodes around the hole. We used the **Spider Leg Nodes** option in the **Element Modify Connectivity** dialog box to select the missing node. (B) shows the modified RBE3 element.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® Element Modify Connectivity** 
Menu	**Edit® Element® Modify Connectivity**

Import and export

Importing a solver input file into an existing FEM or Simulation file

What is it?

For all solvers supported in Advanced Simulation, you can now import a solver input data file into an existing FEM or Simulation file and append the solver data to the existing data.

All FEM and Simulation file entities are imported and appended:

- Nodes
- Elements
- Coordinate systems
- Physical properties
- All other entities, including materials, loads, constraints, regions, degree of freedom sets, solver-specific simulation objects, solver-specific modeling objects, and groups.

To avoid conflicts between the IDs of imported data and existing data, you specify numerical offsets to control how the ID for each type of imported entity is generated. The software generates the default offset for each type of entity by using the next valid ID (the highest ID + 1).

When you import a universal (.unv) file, units will be converted to the units of the target FEM or Simulation file.

This command is not supported for assembly FEMs.

Where do I find it?

Application	Advanced Simulation
Menu	**File® Append**

Nastran support enhancements

Support for surface-to-surface contact

Beginning with this release, **Surface-To-Surface Contact** is available in NX Nastran solution SEBUCKL 105.

For more information, see Surface-to-Surface Contact in the *Advanced Simulation online Help*.

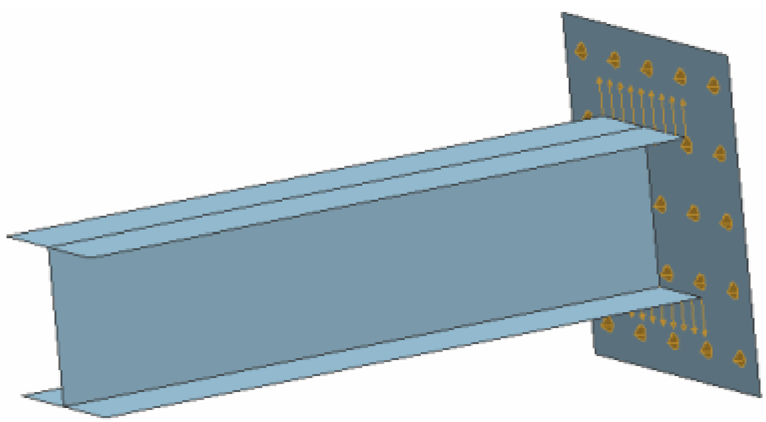
Where do I find it?

Application	Advanced Simulation
Prerequisite	An active NX Nastran solution SEBUCKL 105.
Toolbar	**Advanced Simulation® Surface-to-Surface Contact**
Simulation Navigator	Right-click the **Simulation Object Container® New Simulation Object® Surface-to-Surface Contact**

Edge-to-Surface gluing

What is it?

Use the **Edge-to-Surface Gluing** simulation object to connect an edge to a surface to prevent relative motion in all directions.



To glue an edge to a surface, you must first define the source and target regions where you want to create glue elements. *Glue elements* are stiff springs that connect and constrain the edges and surfaces. A *region* is a collection of element edges or free faces in a section of the model where you expect gluing or contact to occur. You can create the source or edge region using edges of shell or solid elements. You can create the target or surface region using shell elements or free faces of solid elements.

From the **Edge-to-Surface Gluing** dialog box, select a source region and target region in the Simulation model. Specify a search distance and additional glue parameters.

When you solve or export your model, the options in the **Edge-to-Surface Gluing** dialog box define a BGSET bulk data entry in your NX Nastran input file.

Supported analysis types

Edge-to-Surface Gluing is available for all structural NX Nastran solution sequences except for SOL 601 and 701. It is not supported in axisymmetric solutions.

Using a Glue Parameters modeling object to control the glue algorithm

If you include an **Edge-to-Surface Gluing** simulation object in your solution, you can use the options in the **Glue Parameters** dialog box to locally override NX Nastran's global glue parameters. You set global glue parameters using the **Global Glue Parameters** option on the **Case Control** page of the **Solution** dialog box. The glue parameter overrides correspond to the BGPARM bulk data entry in your NX Nastran input file.

For more information, see Glue Parameters overview in the *Advanced Simulation online Help*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file with an NX Nastran solution active
Toolbar	**Advanced Simulation® Edge-to-Surface Gluing**

Creating edge regions

What is it?

When the solution type is NX Nastran, you can now create edge regions and surface regions when you work with glue commands. In previous releases, you only needed to create surface regions when you worked with glue commands.

A new **Edge Region** option is available in the **Simulation Regions** dialog box.

Why should I use it?

You can use the **Edge Region** option to define the source edges for the new **Edge-to-Surface Gluing** command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file with an NX Nastran solution active
Toolbar	**Advanced Simulation**® **Simulation Region** ® **Type** list® **Edge Region**

Enhancements to DDAM event in Response Simulation

What is it?

In NX Response Simulation, when you use the Dynamic Design Analysis Method (DDAM) capability, you can now:

- Specify a DDAM excitation with explicit coefficients for calculating the reference acceleration and reference velocity.
- Change the minimum acceleration (minimum g) value used in the equation that calculates the shock design value of acceleration.

In previous releases, you could only specify multipliers against the standard coefficients (which adhere to the DDS-072-1 specification) and the standard minimum g value was hard-coded at 6 g.

In addition, other design issues have been fixed.

- You can select only a single component direction for the DDAM excitation.
- You can create a DDAM event only if there is a single enforced motion node in the model.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Solved NX Nastran SEMODES 103 – Response Simulation solution and a Response Simulation solution process
Toolbar	**Response Simulation** toolbar® **New Event**

Support for static subcases to define differential stiffness

What is it?

You can now create a static subcase in each of these NX Nastran dynamic solution types:

- SEMODES 103
- SEMFREQ 111
- SEMTRAN 112

The static subcase corresponds to the NX Nastran case control command STATSUB. This case control command selects the static solution to use in forming the differential stiffness for the solution types listed above. For more information, see STATSUB in the *NX Nastran Quick Reference Guide*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file with an NX Nastran 103, 111, or 112 dynamic solution active
Simulation Navigator	Right-click an existing Solution® **New Subcase**

Importing loads as field data

What is it?

If you are working in the Nastran or Abaqus solver environment, the **Import Selective Loads as Field Data** option has been added to the **Import Simulation** dialog box. In previous releases, this option was available as a **Customer Default**.

You can select the **Import Selective Loads as Field Data** option to import certain types of loads, such as Nastran PLOAD2 pressure loads or Abaqus DLOAD distributed loads, as spatial fields

When you import complex models with many different boundary conditions, the **Import Selective Loads as Field Data** option improves performance and usability. Clearing the **Import Selective Loads as Field Data** option may result in slower import processing and a very extensive listing of boundary conditions in the **Simulation Navigator**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation active with Nastran or Abaqus as the specified solver
Menu	**File® Import® Simulation**

Import and export support enhancements

What is it?

This release includes a number of improvements to the import and export support for Nastran input files.

Selective import and export

You now have more granular control over the Nastran input file syntax that NX imports and exports.

- When you import a Nastran input file, you can use the new **Selective Import** options in the **Import Simulation** dialog box to control which bulk data entries that NX imports. You can choose to selectively import the entries either by their **By card name** or **By card family** (such as loads, boundary conditions, or DOFs).

  If you choose the **By card name** option, you can also choose to import selected entries as either commented or uncommented **User Defined Text**. Importing selected bulk data entries as uncommented **User Defined Text** can be helpful if you need to import an entry that is only partially supported for import (not all fields are supported). If you import the entry as uncommented **User Defined Text**, NX imports all fields.

  Note With this capability, do not to make modifications to the unsupported commands or bulk data entries. If you make changes to unsupported syntax, Nastran may either fail to solve the file with a FATAL error message or generate incorrect results.

- When you export a Nastran input file, you can use the new **By card name** option in the **Filter Method** list to selectively exclude certain bulk data entries from export.

Auxiliary file import

In the **Import Simulation** dialog box, when you import an ASCII input file, you can use the new **Auxiliary File Import** support option to import a second ASCII format file. NX includes the auxiliary input file with the primary input file. For example, if you have a group of parameters that you typically use for all analyses, you could save those parameters in a separate file. You could then use the **Auxiliary File Import** option to include that parameter file each time you import a .dat file.

Control over the import of unsupported syntax

If your Nastran input file contains syntax that is not supported in NX, you can use the new **Import unsupported cards** option in the **Import Simulation** dialog box to control whether the software imports that syntax as a commented or uncommented **User Defined Text** modeling object.

For example, you can use this option if your model contains an element, such as the RTRPLT bulk data entry, that is not supported in NX. If you import the element as an uncommented **User Defined Text** modeling object and then solve in NX, Nastran should solve the file and produce correct results.

Note With this capability, do not to make modifications to the unsupported commands or bulk data entries. If you make changes to unsupported syntax, Nastran may either fail to solve the file with a FATAL error message or generate incorrect results.

Case control and bulk data entry support enhancements

The following table lists the support improvements for Nastran bulk data entries and case control commands.

Name	NX Nastran import/export support	MSC Nastran import/export support	Notes
BOLTLD	Yes	No	
CFAST	Yes	No	See **Unsatisfied xref title** When you import from an .op2 file, PIDA and PIDB are not supported.
CREEP	Yes	No	
CWELD	Yes	No	See **Unsatisfied xref title** When you import from an .op2 file, PIDA and PIDB are not supported.
EIGB	Yes	Yes	Only METHOD=SINV is currently supported.
MAT11	Yes	No	See **Unsatisfied xref title** for more information.
MATSMA	Yes	No	

MBDEXPORT Case Control command	Yes	Yes	
PFAST	Yes	No	The following fields are not currently supported: • GE • MCID=-1 • MFLAG See **Unsatisfied xref title** for more information.
PWELD	Yes	Yes	The following fields are not currently supported: • LDMIN • LDMAX • MCID=-1 • MSET • TYPE See **Unsatisfied xref title** for more information.
SKIPOFF Case Control command	Yes (for import only)	Yes (for import only)	
SKIPON Case Control command	Yes (for import only)	Yes (for import only)	
SWLDPRM	Export support only for NX Nastran	No	Only the PRTSW field is currently supported. See **Unsatisfied xref title** for more information.
TABLED2	Yes	Yes	In general, NX converts TABLED2 entries to TABLED1 entries on import. In NX, the value for the independent variable can only be specified once. NX does not convert TABLED2 entries to TABLED1 entries if: • The TABLED2 entry is used in a SOL 601 or SOL 701 solution. • The TABLED2 entry is applied to an SPCD entry.

TABLED3	Yes	Yes	In general, NX converts TABLED2 entries to TABLED1 entries on import. In NX, the value for the independent variable can only be specified once. NX does not convert TABLED2 entries to TABLED1 entries if the TABLED2 entry is used in a SOL 601 or SOL 701 solution.
TABLEST	Yes	Yes	
TEMPD	Yes	Yes	Currently: • NX only supports a single TEMPD entry per solution, not per subcase. • NX expands the TEMPD entry into TEMP bulk data entries on import.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Import® Simulation** **File® Export® Simulation**

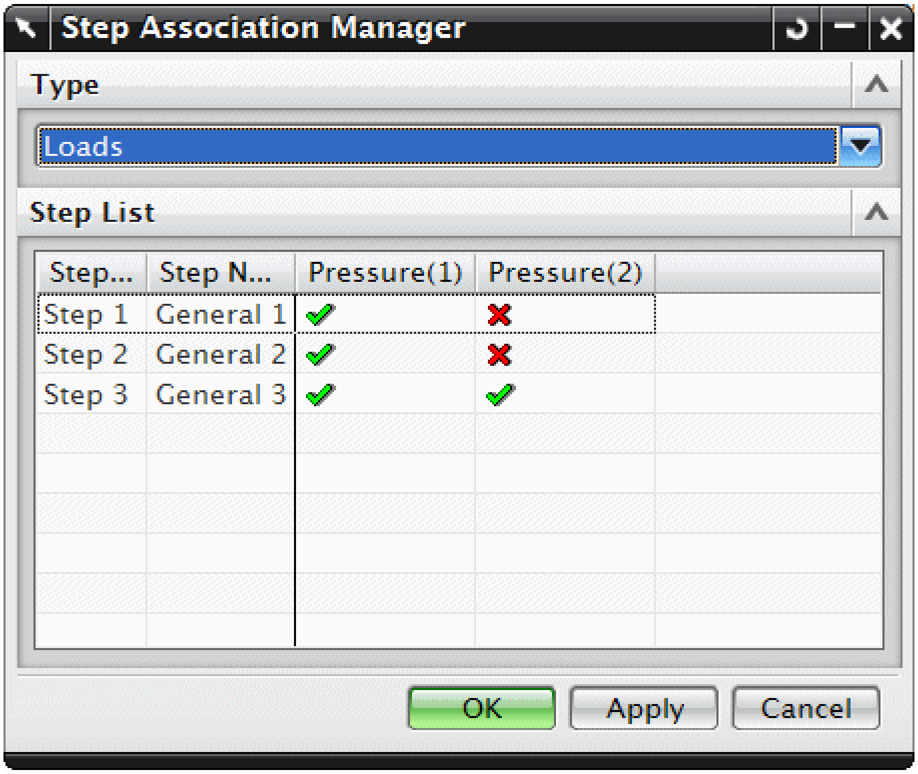
Abaqus support enhancements

Assigning boundary conditions to individual steps

What is it?

When Abaqus is the selected solver, use the new **Step Manager** command if you want to use a table to select the loads, constraints, and simulation objects assigned to each step in the Abaqus solution. After you make the changes, the load, constraint, and simulation object assignments are displayed in the **Simulation Navigator**. As in previous releases, you can continue to assign loads, constraints, and simulation objects using the **Simulation Navigator**.

From the **Step Manager**, you can export the boundary condition settings to a spreadsheet or a browser window.



Solution-level simulation objects

In previous releases, you could assign simulation objects only to the solution. In this release, you can assign simulation objects to either the solution or to steps.

Boundary conditions that are assigned to the entire solution, rather than to steps, do not appear in the **Step Manager**. These boundary conditions include **Automatic Coupling**, **Manual Coupling**, **Tie Surface**, and **Surface-based Coupling** in a General solution.

Step-level simulation objects

In this release, the following simulation objects are now assigned to steps:

- **Surface-to-Surface Contact**
- **Contact with Clearance**
- **Bolt Contact with Clearance**
- **Surface-to-Surface Thermal Conductance**
- **Element Removal** (new in NX 7.5.2)

You can use the **Step Manager** to select or deselect the simulation objects for each step.

If you migrate a model from a previous release, the simulation objects for contact and thermal conductance are written to each step.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with Abaqus as the specified solver
Simulation Navigator	Right-click an existing Solution® **Step Manager**

Removing elements from an analysis step

What is it?

During an Abaqus structural analysis, you may need to add or remove portions of a structure from the finite element model. Use the **Element Removal** simulation object to selectively remove elements from one or more steps in the analysis. In subsequent steps, you can use the **Element Removal** command to reactivate elements, either with or without strain. You can use this simulation object only in General Analysis steps.

The **Element Removal** simulation object corresponds to the `*MODEL CHANGE` keyword.

Why should I use it?

The ability to deactivate and reactivate elements is useful, for example, for analyzing excavation and construction of tunnels.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Abaqus General Analysis solution
Toolbar	**Advanced Simulation® Element Removal** 
Simulation Navigator	Right-click the **Simulation Objects** node under a step® **New Simulation Object® Element Removal**

Changing friction properties

What is it?

In a multi-step Abaqus solution, you can now change the friction properties of contact pair interactions and change the friction definition for each solution step.

Use the new **Abaqus Change Friction Definition** modeling object to change the friction properties of contact pair interactions. You can define:

- Surface interaction (the contact pair)
- Amplitude
- Friction parameters

Abaqus Change Friction Definition corresponds to the parameters for the `*CHANGE FRICTION` and `*FRICTION` keywords.

From the **Solution Step** dialog box, you can change the friction definition for each solution step.

Where do I find it?

To change the friction properties of contact pair interactions:

Application	Advanced Simulation
Prerequisite	An active Abaqus solution
Toolbar	**Advanced Simulation**® **Modeling Objects** ® **Type** list® **Abaqus Change Friction Definition**
Menu	**Insert**® **Modeling Objects**® **Type** list® **Abaqus Change Friction Definition**

To change the friction definition at a solution step:

Application	Advanced Simulation
Prerequisite	An active Abaqus solution with a **General** solution step
Simulation Navigator	Right-click the General solution step® **Edit**® **Change Friction** tab® **Create Change Friction** ® **Create**

New solution control parameters

Use the new **Abaqus Step Control Parameters** type of modeling object to control:

- The convergence control in a nonlinear solution.
- The time incrementation scheme.

The options in the **Abaqus Step Control Parameters** dialog box correspond to the parameters for the `*CONTROLS` keyword. For example, you can use the **Set Analysis=Discontinuous** option to set parameters that generally improve efficiency for severely discontinuous behavior, such as frictional sliding or concrete cracking. The **Set Analysis=Discontinuous** option allows the solver to complete a number of iterations before it begins any checks on the convergence rate.

In most nonlinear analyses, you should not need to adjust the options in the **Abaqus Step Control Parameters** dialog box. However, if your solution contains extreme nonlinearities, you may need to adjust some of these options.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Abaqus solution
Toolbar	**Advanced Simulation® Modeling Objects** ® **Type** list® **Abaqus Step Control Parameters**
Menu	**Insert® Modeling Objects® Type** list® **Abaqus Step Control Parameters**

Importing loads as field data

What is it?

If you are working in the Nastran or Abaqus solver environment, the **Import Selective Loads as Field Data** option has been added to the **Import Simulation** dialog box. In previous releases, this option was available as a **Customer Default**.

You can select the **Import Selective Loads as Field Data** option to import certain types of loads, such as Nastran PLOAD2 pressure loads or Abaqus DLOAD distributed loads, as spatial fields

When you import complex models with many different boundary conditions, the **Import Selective Loads as Field Data** option improves performance and usability. Clearing the **Import Selective Loads as Field Data** option may result in slower import processing and a very extensive listing of boundary conditions in the **Simulation Navigator**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation active with Nastran or Abaqus as the specified solver
Menu	**File® Import® Simulation**

Import and export support enhancements

What is it?

This release includes improved import and export support for a number of Abaqus keywords.

Keyword support enhancements

This release includes support for a number of new Abaqus keywords.

Keyword	Supported parameters	Import support	Export support	Notes
*CHANGE FRICTION	All except ELEMENT	No	Yes	For more information, see **Unsatisfied xref title**.
*CONTACT INTERFERENCE	All	Yes	Yes	*CONTACT INTERFERENCE was only supported for export in previous releases
*CONTROLS	All	No	Yes	For more information, see **Unsatisfied xref title**.
*COUPLING	All	Yes	Yes	*COUPLING was only supported for export in previous releases
*CREEP	All	No	Yes	For more information, see **Unsatisfied xref title**.
*DISTRIBUTING	All	Yes	Yes	*DISTRIBUTING was only supported for export in previous releases
*MODEL CHANGE	TYPE=ELEMENT TYPE=CONTACT PAIR	No	Yes	For more information, see **Unsatisfied xref title**.
*OUTPUT	All parameters except OP	Yes (partial)	Yes	
*SECTION POINTS	All	Yes	Yes	*SECTION POINTS is used in conjunction with the *BEAM GENERAL SECTION keyword. It locates the points in the beam section for which stress and strain output are required.
*TIE	All	Yes	Yes	*TIE was only supported for export in previous releases
*VISCOELASTIC	All	No	Yes	For more information, see **Unsatisfied xref title**.

Support for temperature-dependent stress strain data

Temperature-dependent stress-strain curve data is now supported for both import and export.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

ANSYS support enhancements

Import support improvements

What is it?

This release includes improvements to the import support for ANSYS input files.

Import support for nodal thickness values on 2D elements

When you import an ANSYS input file that contains a 2D mesh, NX now imports the individual nodal thickness values associated with that mesh. In ANSYS, you define nodal thickness values with real constants that have four thickness values for each element. In previous releases, NX computed the average thickness value for each 2D element in an ANSYS input file. Now, NX imports the nodal thickness values for each 2D element. If the nodal thickness values for an element vary, NX imports the individual values as **Element Associated Data**.

Importing material orientation vectors as spatial fields

You can now control how the software imports material orientation vectors for solid elements in the ANSYS solver environment. In the **Import Simulation** dialog box:

- Select the **Import 3D MOVs as field data** check box to import any material orientation vector data for solid elements as a single spatial field.

- Clear the **Import 3D MOVs as field data** check box to import material orientation vector data for solid elements as physical property table data. In NX, there is a one to one relationship between physical property tables and mesh collectors. If your model contains a large number of material orientation vector physical property definitions, the software creates a large number of corresponding mesh collectors in the **Simulation Navigator**. This can make your model difficult to manage.

Note If a coordinate system is only used in a solver input file to provide the orientation for the 3D material orientation vector, the software does not import the coordinate system into NX. This ensures that the software does not import numerous coordinate systems that are not required for the analysis.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with ANSYS as the specified solver
Menu	**File® Import® Simulation**

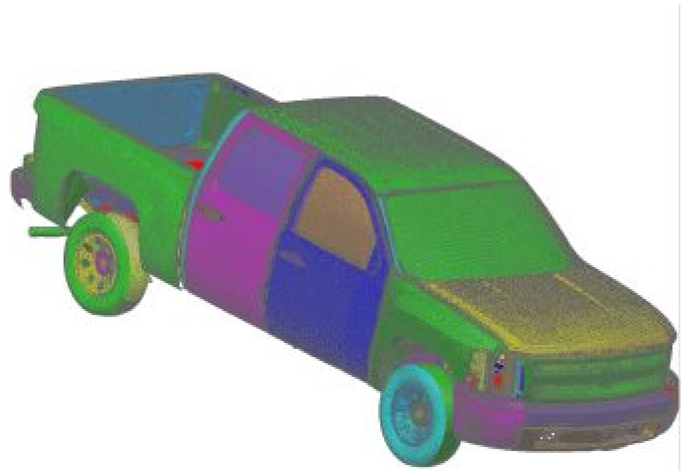
LS-DYNA support enhancements

Basic import support now available

What is it?

NX now offers basic import support for LS-DYNA models. Beginning in this release, you can now import an LS-DYNA keyword file into NX. NX currently supports the import of a subset of LS-DYNA keywords. Keywords supported for import include those for:

- LS-DYNA parts
- Nodes
- Elements (*ELEMENT_INERTIA_, *ELEMENT_DISCRETE, *ELEMENT_SHELL, *ELEMENT_TSHELL)
- Coordinate systems
- Materials (such as isotropic, orthotropic, anisotropic, and *MAT_COMPOSITE_DAMAGE)
- Groups of nodes (*SET_NODE)
- LS-DYNA sections



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with LS-DYNA as the specified solver
Menu	**File® Import**

Import support enhancements

What is it?

This release includes import support for the LS-DYNA keywords listed in the following sections.

Coordinate systems

`*DEFINE_COORDINATE_NODES`

`*DEFINE_COORDINATE_SYSTEM`

`*DEFINE_COORDINATE_VECTOR`

`*DEFINE_SD_ORIENTATION` (imported as the orientation vector for `*ELEMENT_DISCRETE` type elements

Elements

`*ELEMENT_BEAM`

`*ELEMENT_BEAM_OFFSET`

`*ELEMENT_BEAM_ORIENTATION`

`*ELEMENT_BEAM_ORIENTATION_OFFSET`

`*ELEMENT_BEAM_PID`

`*ELEMENT_BEAM_SCALAR`

*ELEMENT_BEAM_THICKNESS

*ELEMENT_DISCRETE (spring-damper)

*ELEMENT_INERTIA

*ELEMENT_INERTIA_OFFSET

*ELEMENT_MASS (non-distributed mass in NX)

*ELEMENT_MASS_NODE_SET (distributed mass in NX)

*ELEMENT_SHELL

*ELEMENT_SHELL_BETA

*ELEMENT_SHELL_MCID

*ELEMENT_SHELL_OFFSET

*ELEMENT_SHELL_THICKNESS

*ELEMENT_SOLID

*ELEMENT_SOLID_DOF

*ELEMENT_SOLID_ORTHO

*ELEMENT_SOLID_TET4TOTET10

*ELEMENT_TSHELL

Groups

The following keywords are imported into NX as groups of nodes:

*SET_NODE_(_TITLE)

*SET_NODE_COLUMN_(_TITLE)

*SET_NODE_GENERAL_(_TITLE)

*SET_NODE_LIST_(_TITLE)

*SET_NODE_LIST_GENERATE_(_TITLE)

The following keywords are imported into NX as groups of elements:

*SET_BEAM_(_TITLE)

*SET_BEAM_GENERAL_(_TITLE)

*SET_BEAM_GENERATE_(_TITLE)

*SET_DISCRETE_(_TITLE)

*SET_DISCRETE_GENERAL

*SET_DISCRETE_GENERAL_(_TITLE)

*SET_DISCRETE_GENERATE_(_TITLE)

*SET_DISCRETE_GENERATE_TITLE_(_TITLE)

*SET_SHELL_(_TITLE)

*SET_SHELL_COLUMN_(_TITLE)

*SET_SHELL_GENERAL

*SET_SHELL_GENERAL_TITLE_(_TITLE)

*SET_SHELL_LIST_(_TITLE)

*SET_SHELL_LIST_GENERATE_(_TITLE)

*SET_SOLID_(_TITLE)

*SET_SOLID_GENERAL_(_TITLE)

*SET_SOLID_GENERATE_(_TITLE)

*SET_TSHELL_(_TITLE)

*SET_TSHELL_GENERAL_(_TITLE)

*SET_TSHELL_GENERATE_(_TITLE)

Materials

The following materials are imported into NX but are not currently referenced by elements in NX. In NX, elements only refer to the **LSDYNA Material ID**.

- *MAT_001_(_TITLE)
- *MAT_002_(_TITLE)
- *MAT_002_ANIS_(_TITLE)
- *MAT_022_(_TITLE)
- *MAT_ANISOTROPIC_ELASTIC_(_TITLE)
- *MAT_COMPOSITE_DAMAGE_(_TITLE)
- *MAT_ELASTIC_(_TITLE)
- *MAT_ORTHOTROPIC_ELASTIC_(_TITLE)

Nodes

- `*NODE`
- `*NODE_RIGID_SURFACE` (imported into NX as the `*NODE` keyword)
- `*NODE_SCALAR_VALUE` (imported into NX as the `*NODE` keyword)

LS-DYNA Part

In NX, all `*PART` keywords with options are imported as `*PART` in NX, except for `*PART_COMPOSITE`, which is imported as a physical property table in the **Advanced Laminates** module.

Spring constants for the *ELEMENT_DISCRETE keyword

These spring constant keywords are imported as modeling objects:

- `*MAT_SPRING_ELASTIC_(_TITLE)`
- `*MAT_S01_(_TITLE)`

Note Options such as _ALE, and _EFG are parsed but are not used in NX.

Section keywords

Section keywords are imported as modeling objects and used in conjunction with the *PART physical property table in NX.

- `*SECTION_BEAM_(_TITLE)`
- `*SECTION_DISCRETE_(_TITLE)` (spring constants for *ELEMENT_DISCTRETE, are imported as Modeling objects)
- `*SECTION_SHELL_(_TITLE)`
- `*SECTION_SOLID_(_TITLE)`
- `*SECTION_TSHELL(_TITLE)`

Note Options such as _ALE, and _EFG are parsed but are not used in NX.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with LS-DYNA as the specified solver
Menu	**File® Import**

Post-processing

Customizable header and color bar

What is it?

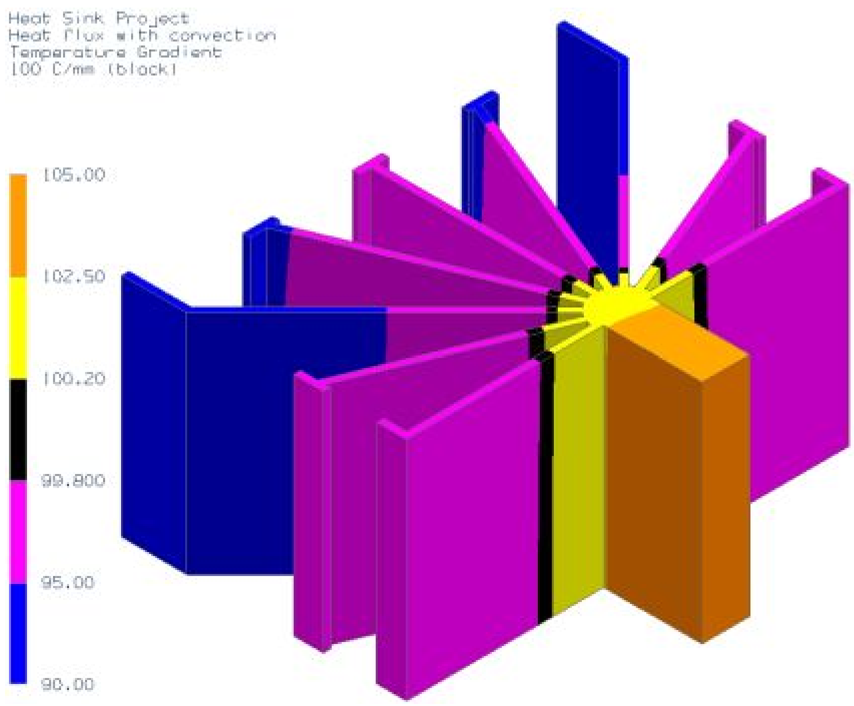
Beginning in this release, you can customize the header text that appears in a post view, and you can assign custom ranges and colors to the color bar.

When customizing the header text, you can:

- Add or remove lines of text.
- Edit the default text.
- Replace text.

When customizing the color bar, you can:

- Modify the range of values assigned to a color band.
- Replace colors to draw attention to a particular range of values or define a custom spectrum.



Why should I use it?

You can customize the header text to be more meaningful to the audience for your analysis, to include project or product information, or to conform to your company's policies and standards.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Results loaded; post view displayed.
Toolbar	**Post Processing**® **Edit Post View**
Menu	**Tools**® **Results**® **Edit Post View**
Location in dialog box	**Post View** dialog box® **Legend** tab

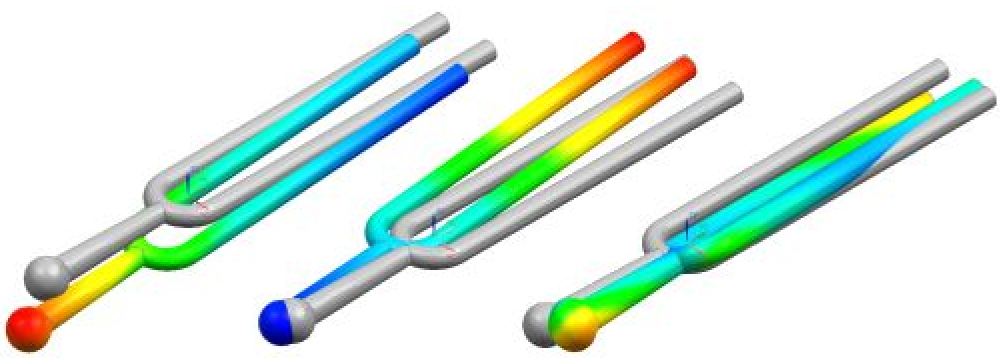
Reference node deformation

What is it?

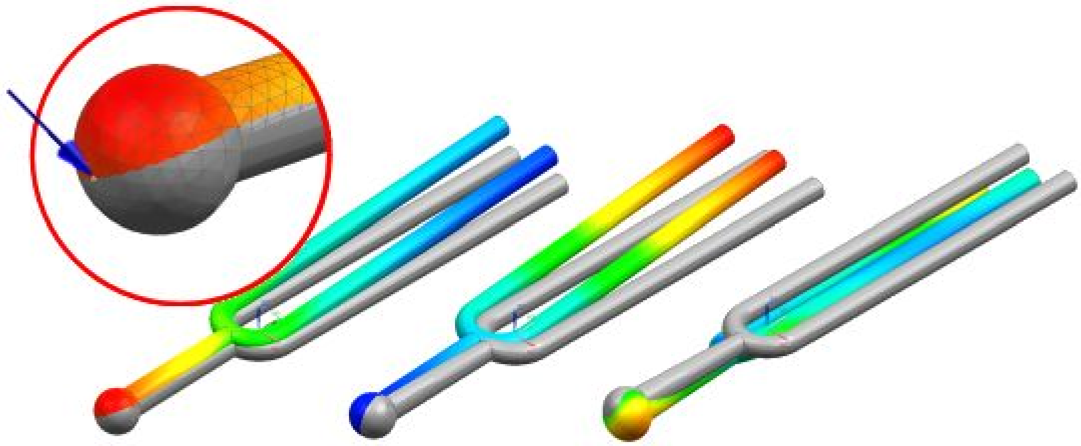
By default, when you create a deformed model display, nodal displacements are transformed and scaled with respect to the origin of the current coordinate system. Beginning in this release, you can transform and scale deformations with respect to a selected node on the model.

You specify a reference node in the **Deformation** dialog box. You can select a node from the model, or enter a node ID in the dialog box.

The following figure shows three rigid body modes transformed with respect to the origin of the absolute rectangular coordinate system. The mode shapes are displayed alongside the undeformed model.



You can select a node on the model to serve as a consistent reference node for deformations. The same three rigid body modes now appear transformed with respect to the selected reference node.



Where do I find it?

Application	Advanced Simulation
Prerequisite	Results loaded; post view displayed.
Toolbar	**Post Processing**®  **Edit**
Menu	**Tools**® **Results**® **Edit Post View**
Post-Processing Navigator	Right-click a post view® **Set Deformation**
Location in dialog box	**Post View** dialog box® **Display** tab® **Deformation**® **Result**® **Reference Node**

Results manipulation

What is it?

Use **Results Manipulation** to create new results by performing operations on one or more existing results sets. Choose from two methods for manipulating results:

- **Envelope** compares two or more results of the same type and component, and returns the minimum or maximum value as a unitless, scalar quantity. The selected components may belong to different load cases, iterations, or solutions, but must be of the same result type (displacement or stress) and be defined at the same output location (nodal, elemental, or element-nodal).

- **Combination** uses standard NX expression syntax to perform operations on one or more results of the same type. The selected results may belong to different load cases, iterations, or solutions, but must be the same in terms of result type, output location, and solution type (either real or complex).

Both methods collect results across the selected types or components based on element or node IDs. Therefore, the underlying FE models must be compatible in terms of the number of nodes and elements, IDs, and element types.

New results are saved as universal (.unv) files. Optionally, you can include model data (nodes and elements) in addition to results values. You can automatically import results into NX for further post-processing or save them as a companion result to an existing solution.

Where do I find it?

Application	Advanced Simulation
Prerequisite	One or more solution results loaded or imported.
Toolbar	**Post Processing®** **Results Manipulation**
Menu	**Tools® Results® Results Manipulation**

Durability

BWI fatigue life criterion

What is it?

The **BWI** fatigue life criterion predicts the fatigue failure in welded joints. This criterion takes into account local stress concentrations which may be due to the weld itself. The weld equation relates the number of stress cycles until failure occurs to the stress range of a stress cycle for a particular weld class.

BWI is based on the British Welding Institute's (BWI) formulation which uses the weld class as the basis for the fatigue life estimate. Because the BWI weld classes in NX are defined by the British Standard BS 5400, you need to be familiar with the weld class definitions to use the BWI fatigue life criterion accurately.

The weld criterion does not use the material properties. You need to specify:

- The weld class in the **BWI Weld Class** list.
- The number of standard deviations below the mean value of the stress range in the **Number of Standard Deviations** field.

The **BWI** fatigue life criterion is recommended for both high-cycle and low-cycle applications.

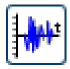
This criterion can also:

- Model the mean stress effect using a method that you specify in the **Equivalent Stress Method** list.

- Correct for the plate thickness when you select **Use Plate Thickness Correction**. See **Unsatisfied xref title** for more information.

See *BWI fatigue life criterion* in the NX 7.5.2 Maintenance Release for more information.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Static Durability Event**  / **Transient Durability Event** 
Menu	**Insert** ® **Durability** ® **Event** ® **Static / Transient**
Simulation Navigator	Right-click the Durability solution process node ® **New Event** ® **Static / Transient** Right-click the static event or transient event node ® **Edit**
Location in dialog box	**Fatigue** tab ® **Fatigue Life** group ® **Fatigue Life Criterion** list

Dang Van fatigue safety factor output

What is it?

You can now use the new **Dang Van** method to calculate the fatigue safety factor and failure index result sets. The **Dang Van** method is a new high cycle multiaxial fatigue safety factor method that is based on the mesoscopic approach.

The Dang Van method is recommended for high-cycle fatigue.

To use the Dang Van method, you need to specify the following new material properties:

- **Fatigue Limit Strength in Bending**

- **Fatigue Limit Strength in Torsion**

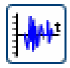
These material properties are available in the **Dang Van Parameters** group on the **Durability** tab of the **Isotropic Material** dialog box

See *Dang Van fatigue safety factor* in the NX 7.5.2 Maintenance Release for more information.

Why should I use it?

Use the Dang Van fatigue safety factor output to determine whether fatigue load paths cause crack initiation.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Static Durability Event**  / **Transient Durability Event** 
Menu	**Insert** ® **Durability** ® **Event** ® **Static / Transient**
Simulation Navigator	Right-click the Durability solution process node ® **New Event** ® **Static / Transient** Right-click the static event or transient event node ® **Edit**
Location in dialog box	**Fatigue** tab ® **Fatigue Safety Factor** group ® **Fatigue Safety Factor Output** ® **Output** list

Thickness correction for BWI and Stress Life criteria

What is it?

The new **Use Plate Thickness Correction** option lets you correct the stress amplitude or stress range for the plate thickness in the following fatigue life criteria:

- **BWI**

- **Stress Life**

To correct for the plate thickness, you need to specify the values for the **Plate Thickness Ratio** and **Plate Thickness Exponent**. See *Plate thickness correction* in the NX 7.5.2 Maintenance Release for more information.

Why should I use it?

In general, the fatigue strength of welds decreases as the joint thickness increases. This effect is more pronounced for welds that are oriented transverse to the applied stress direction. Use the plate thickness correction to account for this effect.

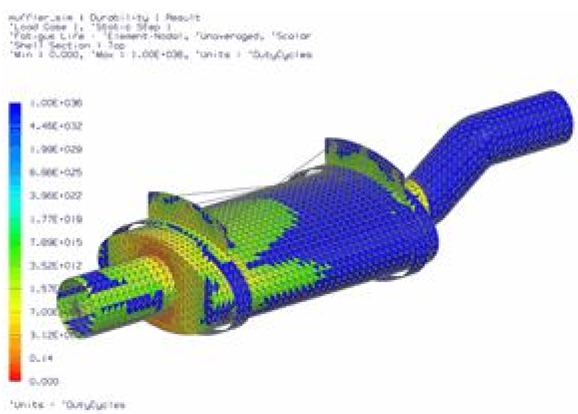
Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Static Durability Event** / **Transient Durability Event**
Menu	**Insert** ® **Durability** ® **Event** ® **Static / Transient**
Simulation Navigator	Right-click the Durability solution process node ® **New Event** ® **Static / Transient** Right-click the static event or transient event node ® **Edit**
Location in dialog box	**Fatigue** tab ® **Fatigue Life** group ® **Plate Thickness Correction** subgroup

Advanced Durability tutorial

What is it?

A getting started tutorial is now available to help you learn how to use the functionality of Advanced Durability. See *Getting started with the Durability solution process* to access the activity in the NX 7.5.2 Maintenance Release.



NX FE Model Correlation and NX FE Model Updating

Export to UNV overview

What is it?

Use the **Export to UNV** command to export the active sensor selection configuration to a UNV file. The UNV file also contains coordinate systems and node coordinates.

The following table lists the UNV datasets that are written.

UNV dataset number	Content	Additional information
151	Header	
164	Units	
2400	Model Header	
2420	Coordinate Systems	The origin is always written in SI units.
2411	Nodes	Nodes are written in the unit system defined by dataset 164. Node values are in double precision.
1802	Coordinate Trace	Coordinate traces contain the active sensor directions.

Why should I use it?

Use this command to export the sensor locations to an UNV file that you can use with a third-party software to obtain experimental modal test data.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active sensor selection configuration
Simulation Navigator	Right-click the Pre-Test Planning solution process node ® **Export to UNV**

Automatic node map size reduction

What is it?

An **Allow Size Reduction** option is now available in the **Correlation** dialog box. Use this option to limit the size of the node map that is created between the work solution and the reference solution by the Correlation solution process.

You specify the node map size in the **Size Limit** option that becomes available when you select the **Allow Size Reduction** check box.

NX reduces the node map to the specified size by dropping paired nodes. Only nodes that are in close proximity to others are recursively dropped in order to maintain a good overall coverage of the entire correlation domain. Nodes that are part of the work solution's **Analysis Set (ASET)** or **Degree of Freedom Set (USET,U2)** are never dropped. This ensures that any nodes important to you are always retained.

The final node map size may be smaller than the specified **Size Limit** value when:

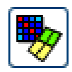
- The reference model has fewer nodes than the specified size.
- NX finds fewer node pairs than the specified size due to the node matching tolerance.

Why should I use it?

You can reduce memory requirements and the computation speed of the correlation metrics, when you correlate two solutions that have large-scale meshes with a large number of modes and paired nodes.

You can also create a matching DOF set from the resulting **Node Map** node when correlating the same analysis solution. The created DOF set has the number of nodes specified by the **Size Limit** value, and is evenly distributed across the model.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Correlation** ® **New Correlation** 
Menu	**Insert** ® **Correlation** ® **New Correlation**
Simulation Navigator	Right-click the Correlation solution process node ® **Edit**
Location in dialog box	**Node Map Settings** group ® **Allow Size Reduction** check box

Mode shape data handling improvements

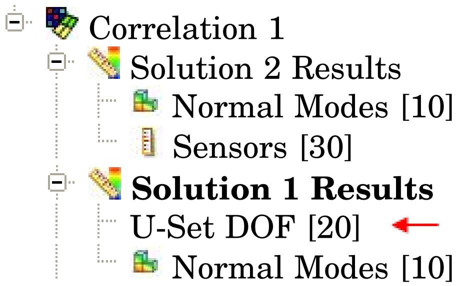
What is it?

The Pre-Test, Correlation, and Model Update solution processes now handle large sets of mode shape data more efficiently. NX defers the loading of mode shape data until it performs the needed computation. Only data at nodes that are part of the correlation domain or of a pre-test DOF set are loaded. The data is then cached in memory for subsequent computations.

U-Set DOF node supported for SEMODES 103 solution

What is it?

The **U-Set DOF [#]** node is now also supported for NX Nastran SEMODES 103 solution. This node is displayed in the **Simulation Navigator** under the work solution results node when an **Degree of Freedom (USET,U2)** DOF set exists in the OP2 result file.



The **U-Set DOF [#]** node is shown for information only. U-Set DOFs do not affect correlation results, but they are taken into account for the node map size reduction. See **Title not found** for more information.

Correlation and Model Update related solution sequences supported in NX Nastran 7.1

What is it?

NX Nastran version 7.1 or later now executes the following solution sequences on all NX Nastran architectures:

- DESOPT 200 - Model Update solution sequence

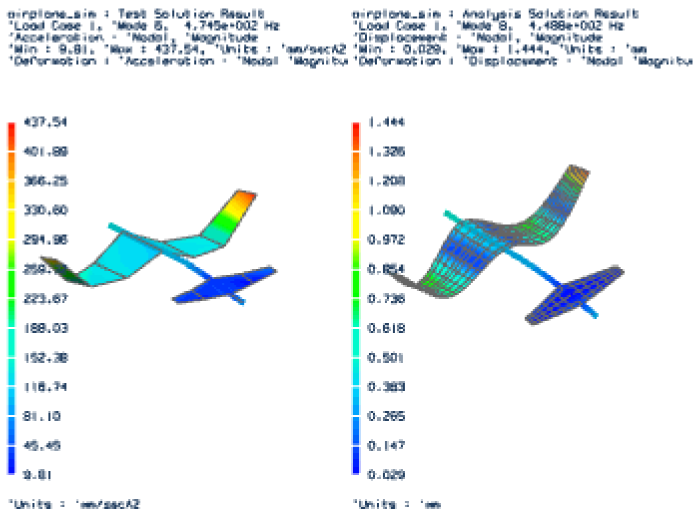
- The optional reduced system matrices computation for the SEMODES 103 solution sequence

In previous releases, NX had to use a DMAP alter in order for Nastran to execute these solution sequences. Because the DMAP alter is no longer required, NX ignores the **Model Update Database Generation** and **Correlation Database Generation** options on the **General** tab in the **Solution** dialog box.

Correlation tutorial

What is it?

A getting started tutorial is now available to help you learn how to use the Correlation solution process in NX FE Model Correlation. See *Getting started with the Correlation solution process* to access the activity in the NX 7.5.2 Maintenance Release.



NX Laminate Composites

Fill Laminate overview

What is it?

Use the **Fill Laminate** command to create a 3D laminate between a set of ANSYS 2D dependent meshes.

The global layup must be defined on the master mesh of the 2D dependent meshes.

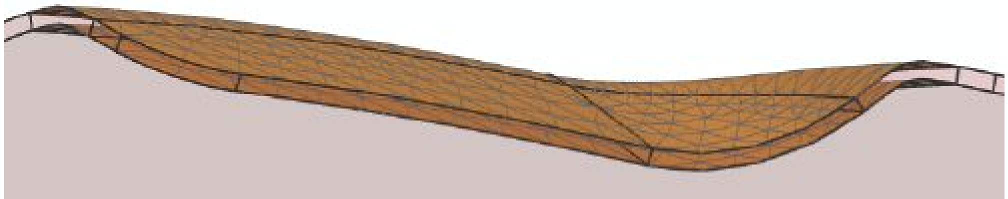
NX creates the following:

- A 3D mesh collector, for each 2D mesh collector referred to by the filling operation, that contains meshes for the 3D plies. The 3D mesh collectors point to laminate physical properties, with stacking recipes set to **Inherit from Layup**.

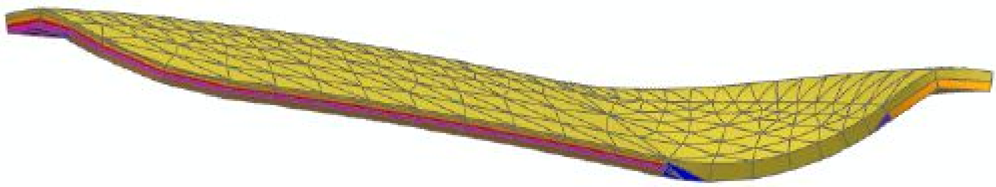
- One mesh collector that contains ply drop-off meshes, and is called **Extrusion Resin**. The **Extrusion Resin** collector points to a solid property and to the isotropic Epoxy material from the NX library. You can change this material.

- One 3D mesh, for each combination of ply and 2D mesh collector, that contains bricks and wedges.

- Drop-off meshes that contain combinations of wedge, tetrahedron, and pyramid elements. These meshes are called **Resin Elements**.



2D dependent meshes and transparent polygon body



3D laminate created between the set of 2D dependent meshes

The 3D elements are extruded to fill the void between the two sets of 2D dependent meshes. NX computes the thickness of each layer by taking into account the thickness of the plies of the global layup and the distance between the set of 2D dependent meshes.

Note The **Fill Laminate** command currently creates 3D meshes only in the ANSYS solver environment.

The following 2D ANSYS element types are supported:

- SHELL91 (6)
- SHELL91 (8)
- SHELL99 (6)
- SHELL99 (8)

The 3D mesh consists of SOLID186 elements.

You cannot view the material orientations of the 3D elements. These material orientations are computed by the ANSYS solver using the material orientation of the base 2D mesh.

You cannot view the fiber orientations of the 3D elements.

Warning If the active layup for the base 2D mesh changes, the 3D meshes are not automatically updated. You must delete the 3D meshes, and use the **Fill Laminate** command again.

Why should I use it?

Using solid meshes provides better results in the ply out-of-plane direction.

Where do I find it?

Application	Advanced Simulation
Prerequisite	ANSYS solver environment
Toolbar	**Laminates** ® **Fill Laminate**
Menu	**Insert** ® **Laminate** ® **Fill**

Output laminate physical property table to Nastran's PCOMPG

What is it?

You can now output the laminate physical property table to Nastran's PCOMPG card.

The Nastran's PCOMPG card is similar to Nastran's PCOMP card. In addition to defining PCOMP quantities for the Nastran solver, PCOMPG also defines the global ply ID.

Note You cannot output the laminate physical property table to Nastran's PCOMPG card if the **Stacking Recipe** is set to **Symmetric** or **Symmetric with core** in the **Laminate Modeler** dialog box.

Why should I use it?

When you export the laminate to the PCOMPG card, you can more easily track the results for the same ply across different elements in the **Post Processing Navigator**.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Laminates** toolbar ® **Laminate Physical Property**
Menu	**Insert** ® **Laminate** ® **Physical Property**
Location in dialog box	**Solver Properties** group ® **Output Format** list ® **PCOMPG**

New unidirectional ply material properties

What is it?

In support to the new Puck failure theory, the following properties have been added for the unidirectional type of ply materials:

- **Mean Stress Magnification Factor**

- **Shear Strain Coefficient** is the constant term of the empirical shear correction term that is a function of shear strain.

- **Inclination Parameter +** is the slope of the fracture envelope at zero normal transverse stress, in tension side.

- **Inclination Parameter –** is the slope of the fracture envelope at zero normal transverse stress, in compression side.

These properties are not common material properties and their value must be found with a set of experiments. The default values were determined for an unidirectional ply material of epoxy and carbon fiber with 60% fiber volume fraction.

See **Title not found** for more information.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Laminates** toolbar ® **Ply Materials**
Menu	**Insert** ® **Laminate** ® **Ply Materials**
Location in dialog box	**Type** list ® **Unidirectional** ® **Create** ® **Puck** group

Additional failure theories

What is it?

Two new failure theories are now available in the **Laminate Modeler** dialog box:

- **Puck**

- **LaRC02**

Both theories apply only to ply materials with unidirectional fibers.

Unlike existing failure theories which only have one failure index per theory, these new theories calculate multiple failure indices:

- Fiber failure index

- Matrix failure index

Same consideration as for the failure index applies to margin of safety and strength ratio.

When you graphically post process failure theory results, the enveloping rule that you defined in the **Laminate Graphical Post Report** dialog box are used.

Thus, for example, if you want to display failure index results for a laminate that has the LaRC02 failure theory selected and the enveloping rule is set to **Abs. Max**, the displayed failure index for each element will be the failure index that is the absolute maximum between the fiber failure index at that element and the matrix failure index at that element.

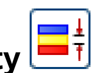
See Puck failure theory and LaRC02 failure theory for more information.

The Puck failure theory needs additional ply material properties that you define in the **Laminate Ply Material** dialog box. See **Title not found** for more information.

Why should I use it?

These theories take into account both the fiber and the matrix so they can determine if the failure occurs due to the fiber or the matrix. The other theories cannot determine that.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Laminates** toolbar ® **Laminate Physical Property** 
Menu	**Insert** ® **Laminate** ® **Physical Property**
Location in dialog box	**Laminate Properties** group ® **Failure Theory** list

FiberSIM ply name in Ply Description field

What is it?

When you import FiberSIM layups using the **Imported Layup** command, NX reads the FiberSIM ply ID from the FiberSIM XML file and stores the IDs in the **Description** cell in the **Layup Modeler** dialog box for each ply in the imported layup.

Why should I use it?

This improvement allows you to know which NX ply corresponds to which FiberSIM ply.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Import at least one FiberSIM layup.
Simulation Navigator	Right-click the layup node ® **Edit**
Location in dialog box	**Description** column

Improved performance for importing plies

What is it?

When you import ply using the **Import Layup** command, NX does it quicker and more efficiently than in the previous releases.

Automatic grouping of layups and global plies

What is it?

You can now create groups that contain elements, polygon faces, or both that are referenced by a given layup or a given ply in a layup.

The following table describes the new right-click commands.

Node	Right-click command	Description
Layups	**Auto-create Layup FE Groups**	Creates one or more groups. Each group contains the elements, polygon faces, or both in one layup and has the name of that layup.
Individual layup node	**Auto-create Layup FE Group**	Creates one group. The group contains the elements, polygon faces, or both in the selected layup and has the name of that layup.
	Auto-create Ply FE Groups	Creates one or more groups. Each group contains the elements, polygon faces, or both in one ply of the selected layup and has the name of that layup followed by the name of that ply.
Individual ply node	**Auto-create Ply FE Group**	Creates one group. The group contains the elements, polygon faces, or both in the selected ply and has the name of ply's corresponding layup followed by the name of that ply.

Where do I find it?

Create one or more element groups, one for each layup.

Application	Advanced Simulation
Simulation Navigator	Right-click the **Layups** node ® **Auto-create Layup FE Groups**

Create one layup element group.

Application	Advanced Simulation
Simulation Navigator	Right-click the individual layup node ® **Auto-create Layup FE Group**

Create one or more element groups, one for each ply in a given layup.

Application	Advanced Simulation
Simulation Navigator	Right-click the individual layup node ® **Auto-create Ply FE Groups**

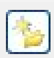
Create one ply element group.

Application	Advanced Simulation
Simulation Navigator	Right-click the individual ply node ® **Auto-create Ply FE Group**

Command name change

What is it?

The following table shows the old and new names of the following right-click commands.

Node	Icon	Old right-click command name	New right-click command name
Zones		**Create Element Groups**	**Auto-create Zone FE Groups**
Name of physical property		**Create Element Groups**	**Auto-create Zone FE Groups**
Individual zone node		**Create Element Groups**	**Auto-create Zone FE Group**

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the **Zones** node or node with the name of the physical property ® **Auto-create Zone FE Groups** Right-click the individual zone node ® **Auto-create Zone FE Group**

NX Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal

Create a board mesh with orthotropic thermal conductivity in a new Simulation file

What is it?

You can now create a Simulation file that also takes into account the orthotropic thermal conductivity behavior of a PCB. This behavior is modeled using:

- Thermal conductivity fields, one each layer of the PCB.

- Thermal resistance fields, one between each layer of the PCB due connections given by vias and dielectric layers.

- **Override Set** simulation objects that replace the mesh thermal conductivity value with the thermal conductivity field values.
  Boards with multiple layers are supported using multi-layer shells, in this case one **Override Set** simulation object is created for each layer of the multi-layer shell mesh.

- **Thermal Coupling** simulation objects that connect each layer of the PCB using the thermal resistance fields.

PCB Exchange uses the board thermal conductivity XML file to create the thermal conductivity and thermal resistance fields. You specify the board thermal conductivity XML file in the **Create ESC Solution** dialog box.

Why should I use it?

You can model your electronic systems more accurately. Complex heat conduction paths exist in PCBs due to dielectrics, traces, vias, and other entities. These paths have a strong impact on the thermal model. These are best represented by discretazing each layer of the PCB and using this information in the thermal mesh.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC

Where do I find it?

Application	PCB Exchange
Prerequisite	An XML *Board Thermal Conductivity* file.
Toolbar	**PCB Exchange** ® **Create ESC Solution**
Menu	**PCB Exchange** ® **Thermal/Flow Simulation**® **Create ESC Solution**

Freeze and reactivate flow field calculation at specified times

What is it?

In addition to the already existing freeze flow conditions, you can now set specified times at which you want to:

- Freeze the flow field calculation. In this case, only energy and scalar equations are solved.
- Reactivate the flow field calculation. In this case, all flow solver equations are solved.

All freeze flow conditions are cumulative.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

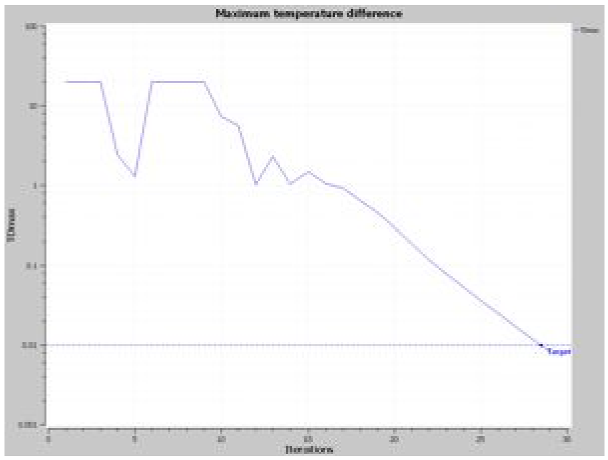
Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solve** ® **Edit Solver Parameters**
Menu	**Analysis** ® **Solve** ® **Edit Solver Parameters**
Simulation Navigator	Right-click the solution node ® **Edit Solver Parameters**
Location in dialog box	**3D Flow Solver** tab ® **Freeze Flow Field for Transient** group ® **Freeze Flow Field at Specified Times** checkbox

Track the convergence of the thermal solver during the solve

What is it?

During the solve, in addition to tracking the convergence of the flow solver in the **Solution Monitor**, you can now also track the convergence of the thermal solver. In a thermal or coupled steady state solution, you can display the maximum temperature difference between two consecutive iterations as a function of iterations.



In a thermal or coupled transient solution, in addition to the previously shown graph, you can also display the number of iterations per time step required for convergence as function of the time.

- The maximum temperature difference as function of iterations in *Maximum temperature difference.png*.
- The number of iterations per time step as function of time in *Iterations vs Time.png*.

Why should I use it?

Use this option to monitor the convergence of the maximum temperature difference between two iterations in your model during the solve.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC Thermal-Flow
NX Space Systems Thermal	Thermal	Space Systems Thermal

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
Toolbar	**Advanced Simulation** toolbar® **Solution**
Menu	**Insert**® **Solution** **Analysis**® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the Simulation file node® **New Solution** Right-click the solution node® **Edit**
Location in dialog box	**Solution Monitor** window® **Graphs**® **Convergence**

Track the results from the full model during the solve

What is it?

During the solve, in the **Solution Monitor**, you can now display the following graphs:

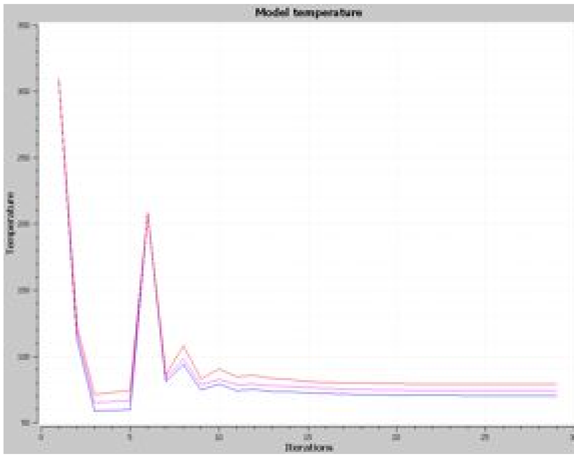
- For a thermal and coupled steady state solution, the solid temperature as function of iterations.

- For a thermal and coupled transient solution, the solid temperature as function of time.

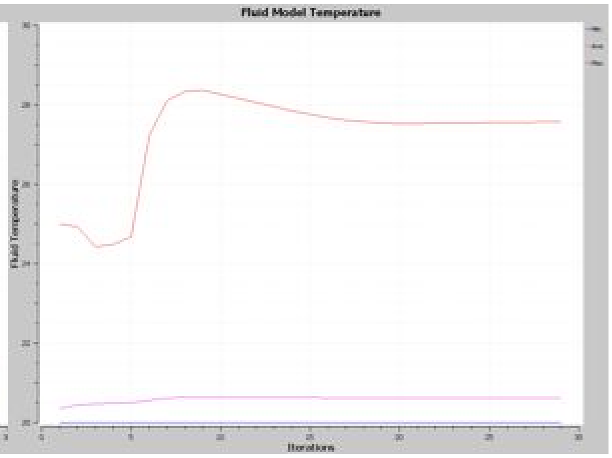
- For a flow and coupled steady state solution:
  - o The fluid temperature as function of iterations
  - o The fluid velocity as function of iterations
  - o The fluid pressure as function of iterations

- For a flow and coupled transient solution:
  - o The fluid temperature as function of time
  - o The fluid velocity as function of time

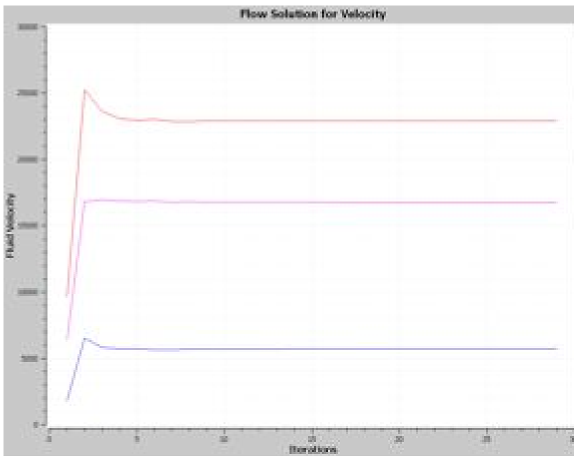
- o The fluid pressure as function of time



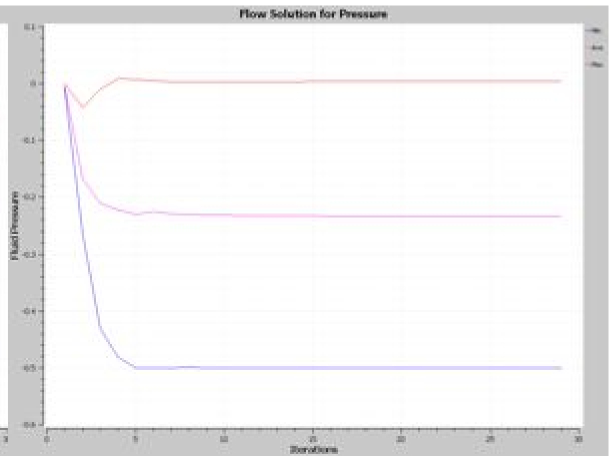
Solid temperature as a function of iterations



Fluid temperature as a function of iterations



Fluid velocity as a function of iterations



Fluid pressure as a function of iterations

After the solve, the software saves the previous graphs in the run directory, in PNG format.

Why should I use it?

Use this option to monitor flow velocity, pressure, and temperature during flow or coupled analysis and solid temperature during thermal or coupled solution.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling 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

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar® **Solution** 
Menu	**Insert**® **Solution** **Analysis**® **Solve**  ® **Edit Solution Attributes**
Simulation Navigator	Right-click the Simulation file node® **New Solution** Right-click the solution node® **Edit**
Location in dialog box	**Solution Monitor** window® **Graphs**® **Track Results**® **Full Model**

Track during solve report

What is it?

You can now track during a thermal or coupled solution, the following temperatures of the thermal region of your model for the selected elements:

- Average temperature as function of time in a transient solution or function of iterations in a steady state solution.

- Minimum temperature as function of time in a transient solution or function of iterations in a steady state solution.

- Maximum temperature as function of time in a transient solution or function of iterations in a steady state solution.

In previous versions, you could only track flow quantities of the flow region of your model.

After the solve, the software saves the graphs and the data in the run directory. The thermal data is stored in *TrackReportThermal.csv* and the flow data is stored in *TrackReportFlow.csv*.

Why should I use it?

Use this option to monitor the temperature during the thermal analysis on selected elements of your model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling 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

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar® **Report**
Simulation Navigator	Right-click the **Simulation Object** container node® **New Simulation Object**® **Report**
Location in dialog box	**Type** list® **Track During Solve**

Select initial condition time step from a previous solution

What is it?

In this release, the usability of defining the time step from another solution as the initial condition for the current solve is improved.

Use the new **Results to Use for Initial Conditions** list in the new **Initial Conditions Selection** group on the **Initial Conditions** tab of the **Solution** dialog box to set the initial condition time step to:

- The start time of the current solution by selecting **At Start Time**.
- Any other time by selecting **At Specified Time** and specify the time in the **Initial Conditions Use Results at Time** box.

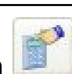
Previously, you needed to specify the start time to be the same as the time step that has the results you want to use as the initial condition.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow
	Axisymmetric Thermal	Axisymmetric Thermal Advanced Axisymmetric Thermal

Where do I find it?

Specify the initial condition time step while creating a solution

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solution** 
Menu	**Insert** ® **Solution**
Simulation Navigator	Right-click the Simulation file node ® **New Solution**
Location in dialog box	**Initial Conditions** tab ® **Initial Conditions** list ® **From Results in Other Directory** or **From File (TEMPF format)**

Specify the initial condition time step while editing an existing solution

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solve** ® **Edit Solution Attributes**
Menu	**Analysis** ® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the solution node ® **Edit**
Location in dialog box	**Initial Conditions** tab ® **Initial Conditions** list ® **From Results in Other Directory** or **From File (TEMPF format)**

Specify duct flow solver options

What is it?

You can now specify the following duct flow solver options in the new **Duct Flow Solver** group on the **Thermal Solver** tab in the **Solver Parameters** dialog box:

- **Pressure Convergence Criteria** box sets the value for the pressure convergence of the duct flow network equations.
- **Iteration Limit** box sets the maximum number of iterations for the hydraulic loop.
- **Relaxation Factor** box sets the damping parameter for the duct network.
- **Convergence Trace** check box defines whether you want a summary of duct network convergence printed for each hydraulic loop iteration.

You can now specify the following duct flow option in the new **Duct Flow Parameters** group on the **Thermal** tab in the **Solution** dialog box:

- **Compressible Flow** check box sets if the flow in the ducts is compressible or not. If the flow is compressible, the density and static pressure calculations take into account compressible effects for all elements with Mach number greater than 0.1.

Why should I use it?

You can now easily define necessary duct flow options in the **Solution** and **Solver Parameters** dialog boxes. Previously you could only define them through **Advanced Parameters** modeling objects.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

Solver Parameters dialog box

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solve** ® **Edit Solver Parameters**
Menu	**Analysis** ® **Solve** ® **Edit Solver Parameters**
Simulation Navigator	Right-click the solution node ® **Edit Solver Parameters**
Location in dialog box	**Thermal Solver** tab ® **Duct Flow Solver** group

Solution dialog box

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solution** **Advanced Simulation** toolbar ® **Solve** ® **Edit Solution Attributes**
Menu	**Insert** ® **Solution** **Analysis** ® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the Simulation file node ® **New Solution** Right-click the solution node ® **Edit**
Location in dialog box	**Thermal** tab ® **Duct Flow Parameters** group

Spatial vector field for orthotropic thermal conductivity override

What is it?

When you specify an override for the orthotropic thermal conductivity, you can now specify a spatial vector field on your model or part of the model.

Previously, you could only specify a spatial scalar field even if the orthotropic thermal conductivity has three components.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling 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

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Override Set — Thermal Properties**
Simulation Navigator	Right-click the **Simulation Object Container** node or the **Simulation Objects** node ® **New Simulation Object** ® **Override Set — Thermal Properties**
Location in dialog box	**Type** list ® **Orthotropic Thermal Conductivity** ® **Distribution** group ® **Method** list ® **Spatial**

Text changes in dialog boxes

What is it?

The following table lists the text changes in dialog boxes.

Dialog box name	Location in the dialog box	Old text	New text
Concentrated Mass	**Properties** group, **Properties** subgroup	**Mass**	**Non-Structural Mass**
Beam	**Properties** group, **Non-Structural Mass** subgroup	**Mass per Length**	**Non-Structural Mass**
Multi-Layer Shell Non-Uniform	**Properties** group, **Stack Definition** subgroup	**Stack Layers**	**Stack Layers (First Layer is Top Layer)**
Solver Parameters	**3D Flow Solver** tab, **Freeze Flow Filed for Transient** group	**Freeze Flow Field**	**Freeze Flow Field Based on Solution Evaluation**
Override Set — Thermal Properties	**Type** ® **Orthotropic Thermal Conductivity**, **Magnitude** group	**Kx** **Ky** **Kz**	**K1** **K2** **K3**
Report	**Type** list	**Track During Flow Solve**	**Track During Solve**

NX 7.5.2 Motion Simulation

Results manipulation in Flexible Body analysis

What is it?

During a **Flexible Body** analysis, after solving for transient results, you can now highlight the step in which the minimum or maximum stress, displacements, or other result types occurred, using the new **Results Manipulation** command. This command lets you create new results by performing operations on one or more existing results sets.

For more information, see **Unsatisfied xref title**.

Where do I find it?

Application	Motion Simulation
Prerequisite	A solved **Flexible Body** solution.
Menu	**Tools® Results® Results Manipulation**

NX 7.5.1 Advanced Simulation

Solver version support

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.

NX7 releases

Solver	File Type	NX 7	NX 7.5	NX 7.5.1
NX Nastran	Import ASCII (.dat)	6.1	7.0	7.0
	Import Binary (.op2)	6.1	7.0	7.0
	Export ASCII (.dat)	6.1	7.0	7.0
	Post-processing of Results	6.1	7.0	7.1
MSC Nastran	Import ASCII (.dat)	2008r1	2008r1	2008r1
	Import Binary (.op2)	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2008r1	2008r1	2008r1
	Post-processing of Results	2008r1	2008r1	2008r1
Abaqus	Import ASCII (.inp)	6.8-1	6.9–1	6.9–1
	Import Binary	N/A	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9	6.9
	Post-processing of Results (.fil)	6.8-EF2	6.9.2	6.9.2
	Post-processing of Results (.odb)	6.8-EF2	6.9–EF1	6.9–EF2
ANSYS	Import ASCII (PREP7, CDB)	12	12.1	12.1
	Import Binary (.rst, .rth)	12	12.1	12.1
	Export ASCII (.inp)	12	12.1	12.1
	Post-processing of Results	12	12.1	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A
	Export ASCII (.k)	971R3.2.1	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A	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

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
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

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
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

Solver	File Type	NX 4	NX 4.0.1	NX 4.0.2	NX 4.0.3	NX 4.0.4
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

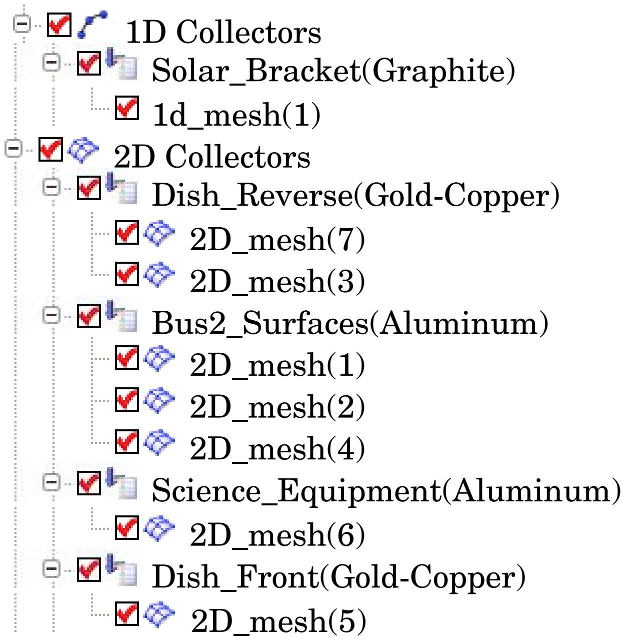
Create groups of meshes automatically from common attributes

What is it?

You can use the new **Automatic Group** command to group your meshes by common attribute. You can group meshes by these attributes:

- Material
- Physical property
- Color
- Beam cross-section
- Laminate physical property
- Element family and mesh type
- Mesh collector type

For example, suppose you have the following 1D and 2D meshes that use different materials and display colors.



Using the **Automatic Group** command, you can group these meshes by material, by color, and by mesh type:

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file.
Simulation Navigator	Right-click the **Groups** node® **Automatic Group**

Simulation Navigator enhancements

What is it?

This release includes several enhancements to the **Simulation Navigator**.

Sort and filter enhancements

You can now sort and filter the contents of the following containers in the **Simulation Navigator**:

- **Polygon Geometry**
- **Mesh collectors**
- **Groups**
- **DOF Sets**
- **Fields**
- **CSYS** (first level of hierarchy only)
- **Regions**
- **Simulation Object Container**
- **Load Container**
- **Constraint Container**

Use the new sort and filter capabilities to manage the display of **Simulation Navigator** data when you work with complex models.

- With the **Sort** command, you can organize the contents of a container by name or type. Within a mesh collector, you can also organize meshes according to the number of nodes or elements in the mesh or according to the element size.

- With the **Filter** command, you can use a wildcard string to limit which contents in the container display in the **Simulation Navigator**. Wildcards are case sensitive.

For example, in the **Load Container**:

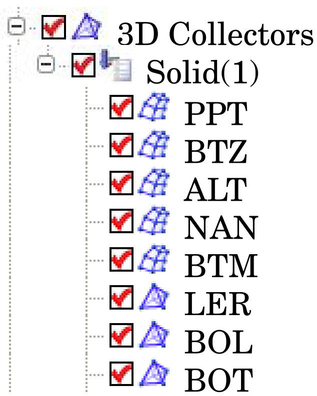
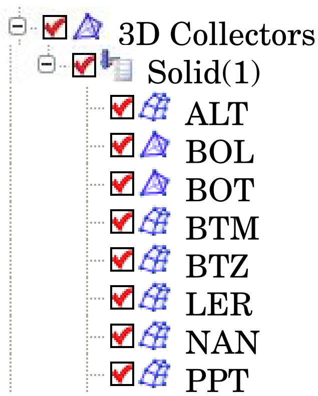
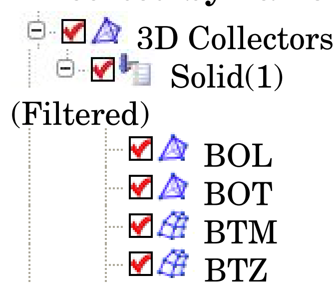
- o To list only the loads with names beginning with the letters *Fo*, enter the wildcard **Fo***.
- o To list only the loads with names ending with the characters *(1)*, enter the wildcard ***(1)**.

In the **Polygon Geometry** container, you can also use the **Filter** command to display only the bodies that are either meshed or unmeshed.

The **Status** column in the **Simulation Navigator** indicates whether the contents of a container are filtered or sorted. When you apply a filter to a selected container, the software appends the string `(Filtered)` to the name of the container.

Note Changes you make to the visibility of data in the **Simulation Navigator** with the **Filter** and **Sort** commands only apply to your current NX session.

The following example shows how the listed contents of the **Solid(1)** mesh collector change depending on the current filter and sort options.

Mesh collectors, Filter and Sort turned off	Mesh collectors sorted by name	Mesh collectors, wildcard filter = B*, sorted by name
		

New icons to indicate different types of bodies

The **Simulation Navigator** now uses icons to help you distinguish between:

-  Solid bodies
-  Sheet bodies
-  Midsurface sheet bodies

These icons help you quickly differentiate between regular sheet bodies and sheet bodies created with the **Midsurface by Face Pairs** or **User Defined Midsurface** commands.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click a supported container® **Filter** or **Sort**

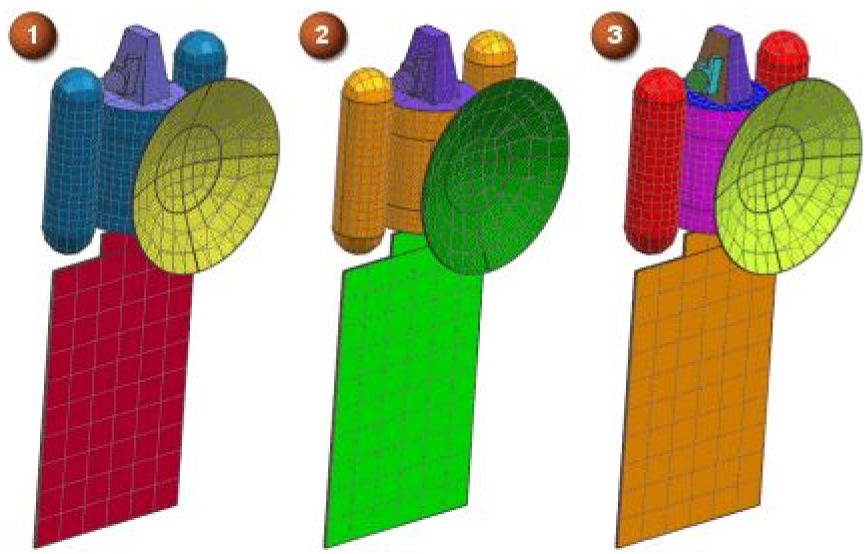
Mesh Color Basis enhancements

What is it?

Two new options have been added to the **Color Basis** model display option that was introduced in NX 7.5. Now, you can define the **Color Basis** by **Mesh Collector** and by **Mesh**.

- The **Mesh Collector** option assigns the same color to all meshes that reference the same mesh collector.
- The **Mesh** option assigns a different color to each mesh. Colors are assigned arbitrarily.

In the **Customer Defaults** dialog box, under **Simulation**® **Extras**, you can change the number of colors to use and you can specify the colors.



(1) Default colors; (2) Colors with Mesh Collector selected; and (3) Colors with Mesh selected

Also, beginning with this release, after you apply the **Color Basis** setting, you can apply the new colors to the meshes permanently by clicking the **Set Mesh Colors** button.

Where do I find it?

Application	Advanced Simulation
Menu	**Preferences®** **Model Display**
Location in dialog box	**Element** tab® **Color Basis** list

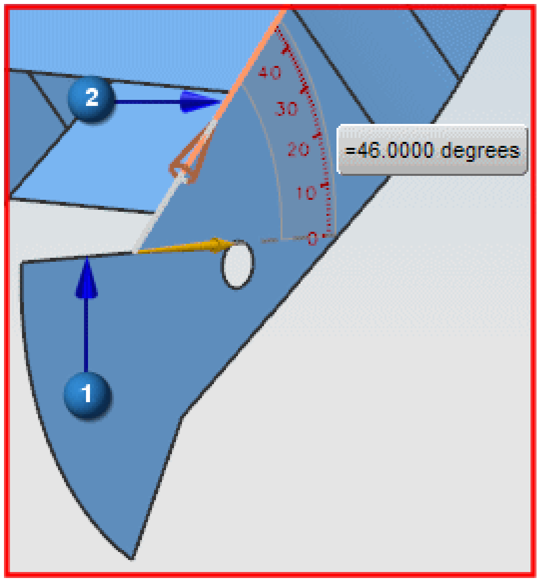
Tangent-continuous edge selection method

What is it?

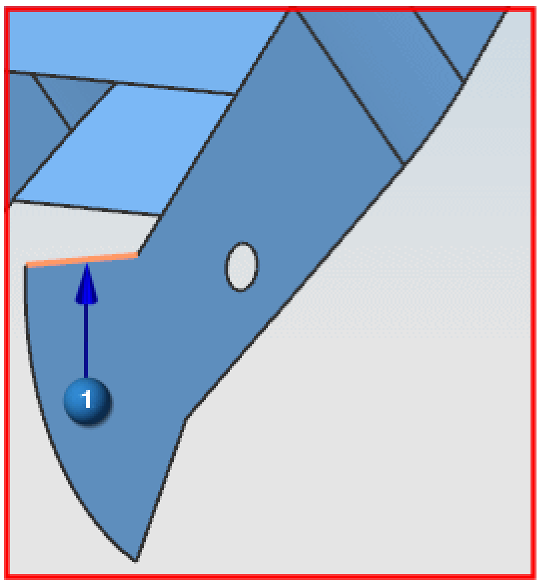
A new selection method named **Tangent Continuous Edges** has been added to let you more easily select polygon edges when you define geometry-based objects such as loads, constraints, and regions (for example, see the **Force** command). This method selects all the edges that are tangent-continuous to the initial edge that you select.

At each vertex of the initial edge you select, the software calculates the tangent vector for all edges that share that vertex. The adjacent edges are considered tangent-continuous if the angle between their tangent vectors is less than the **Tangent Angle Tolerance** specified in the **Smart Selector Options** dialog box.

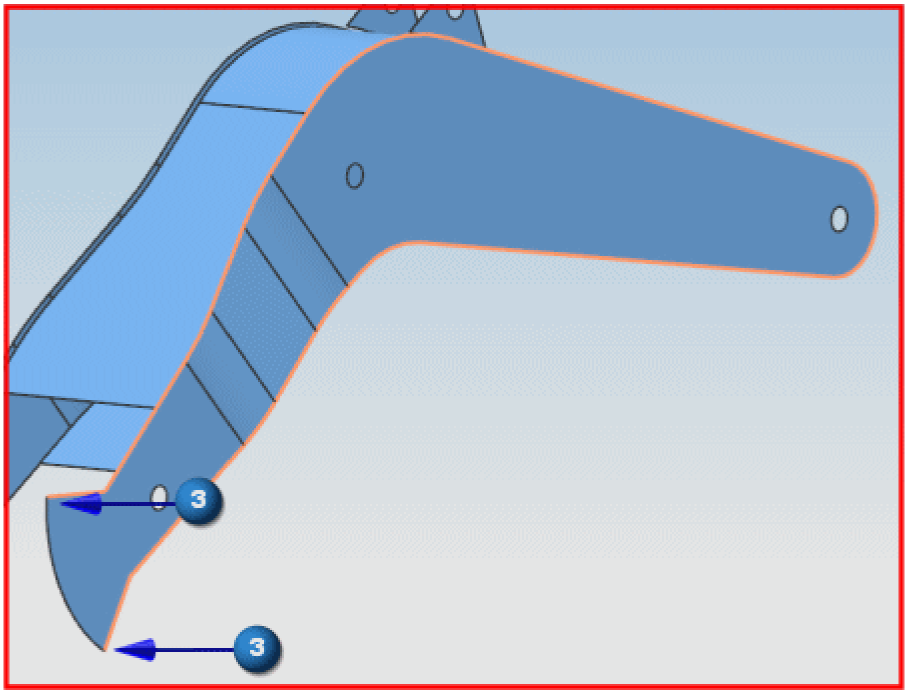
In the following example, the angle between the tangent vectors for edges 1 and 2 is 46 degrees.



With the default **Tangent Angle Tolerance** of 2 degrees, when you select edge 1, only that edge is selected.



However, when you change the **Tangent Angle Tolerance** to **47** degrees, all edges are selected except for the edge that is adjacent to the 87-degree angle at .



Where do I find it?

Application	Advanced Simulation
Toolbar	**Selection bar**® **Method** list® **Tangent Continuous Edges**

Control over default names of polygon bodies

What is it?

This release includes a new **Customer Default** that you can use to control whether the software uses the names of existing assembly components to name the related polygon bodies.

- If you select this default, the software uses the assembly component’s name as the root name for the associated polygon bodies.
- If you do not select this default, the software uses the root name POLYGON BODY for all polygon bodies.

Although you can always later rename polygon bodies with more descriptive strings, you can use the new option to give the bodies more meaningful default names.

To access this option:

1. From the **File** menu, choose **Utilities→Customer Defaults**.

2. In the **Customer Defaults** dialog box, choose **Simulation→Extras**.

3. On the **Miscellaneous** tab, either select or clear the **Assign assembly component names for polygon bodies** option.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file

Midsurface enhancements

Additional display capabilities

What is it?

This release includes two new display options in the **Midsurface by Face Pairs** dialog box.

- Select the **Show Pairing Faces as Transparent** option to have the software display all paired faces as transparent.

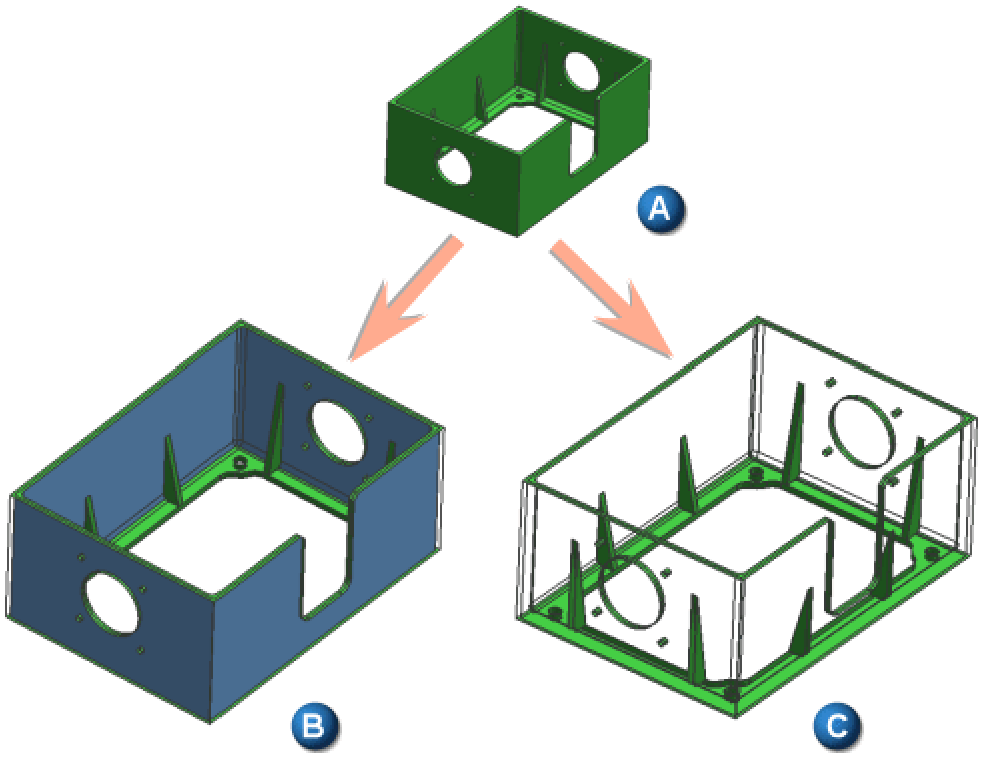
- Select the **Show Midsheets as Transparent** option to have the software display the individual mid-sheets as transparent.

Both of these new options allow you to manage your display while you create and modify face pairs in the **Midsurface by Face Pairs** dialog box. You can use these two options separately or simultaneously to better visualize which faces in your model have been paired. In the following example:

- (A) shows the solid body.

- (B) shows the body after the initial set of face pairs has been created. Here, the **Show Pairing Faces as Transparent** option is selected. The blue faces are the mid-sheets that the software generated for the face pairs. The green faces are faces that are not currently in a pair.

- (C) shows the body with both the **Show Pairing Faces as Transparent** and the **Show Midsheets as Transparent** options selected. Notice how you can easily identify the faces that still need to be paired to complete the midsurface.



These new display options only temporarily change the translucency of the faces in your model. Once you exit the **Midsurface by Face Pairs** dialog box, the software changes the display of the faces and mid-sheets back to shaded.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A part or idealized part displayed
Toolbar	**Midsurface® Midsurface by Face Pairs**

Additional trimming control for replacement mid-sheets

What is it?

When you use the **Replacement Mid-Sheet** option in the **Midsurface by Face Pairs** dialog box, you can now control whether the software trims that replacement mid-sheet. In the **Advanced Trimming Tools** options, if you select the new **Skip Trimming** method, the software does not trim the replacement mid-sheet against the solid body, the faces in the pair, or against the neighboring mid-sheets. Use **Skip Trimming** when you want to use the replacement mid-sheet body without any modifications to its size or shape.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A part or idealized part displayed
Toolbar	**Midsurface® Midsurface by Face Pairs** 

Material and physical properties

Define 3D element material orientation using element faces

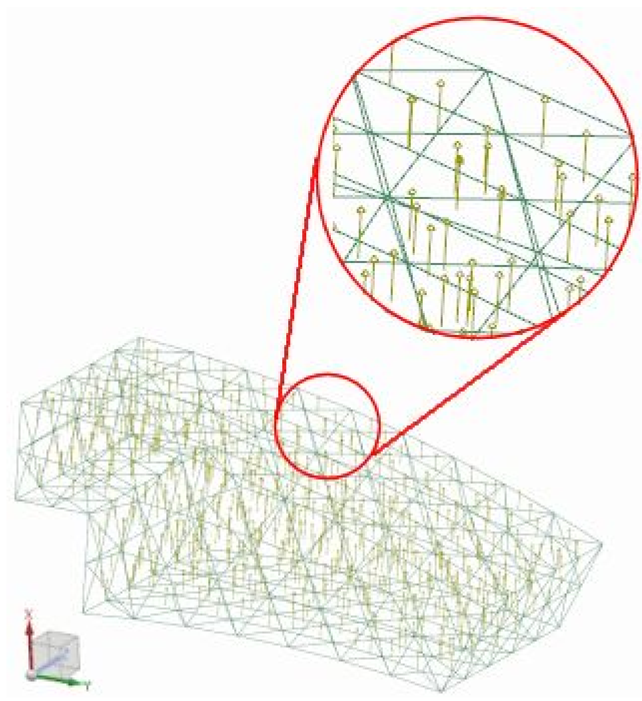
What is it?

For Nastran and Abaqus solvers, the new **Element Free Face Proximity** option has been added to the **Material Orientation Method** list in the **Mesh Associated Data** dialog box.

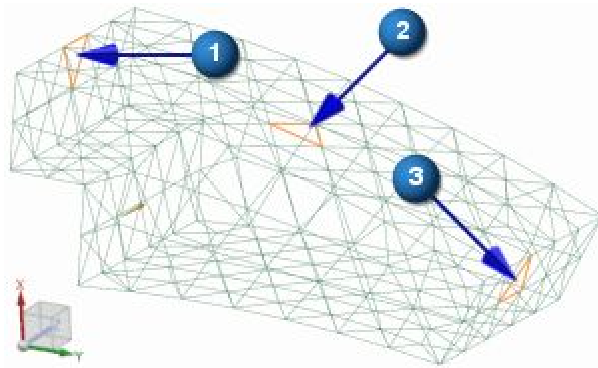
This option lets you define the first direction of the material orientation for a mesh of 3D elements by selecting the free face of one or more of the elements.

- If you select a face in a single element, the software uses the normal of the selected face for all elements in the mesh.
- If you select a face in multiple elements, the software uses the normal of the selected face that is closest to a given element in the mesh.

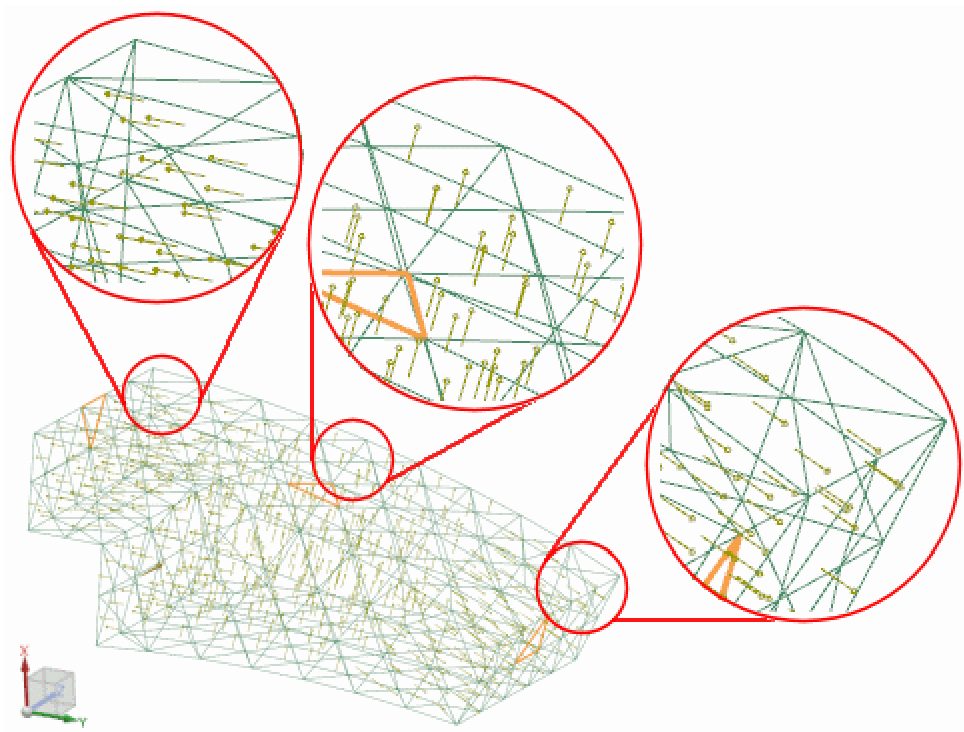
In the following example, the first direction of the current material orientation is in the X direction of the global coordinate system.



Using the new material orientation method, suppose you select element facess (1), (2), and (3):



For each element in the mesh, the resulting first direction of the material orientation depends on the element's proximity to the three element faces that you selected:



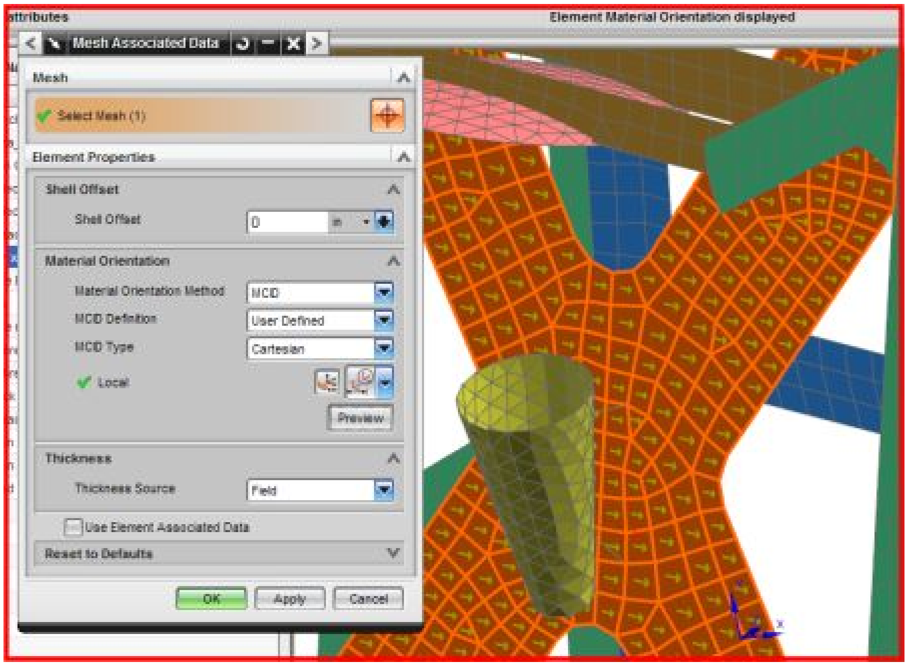
Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file.
Simulation Navigator	Right-click a mesh® **Edit Mesh Associated Data**
Location in dialog box	**Material Orientation Method® Element Free Face Proximity**

Preview material orientation vector graphics

What is it?

A new **Preview** button has been added to the **Mesh Associated Data** dialog box. You can use this button to temporarily display the material orientation vector graphics on the elements before you apply material orientation changes in a mesh.



You can edit the colors and displayed directions of the previewed material orientation vector graphics using the **Element Material Orientation** options in the **Finite Element Model Check** dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file.
Simulation Navigator	Right-click a mesh® **Edit Mesh Associated Data**
Location in dialog box	**Preview** button

New workflow for editing attributes of multiple meshes

What is it?

The workflow for editing the attributes of multiple meshes has changed. The **Mesh Associated Data** dialog box now includes icons that indicate whether the attribute values of the selected meshes are either mixed or identical. You can then apply new attribute values to all selected meshes simultaneously.

For more information, see Edit the mesh associated data for multiple meshes.

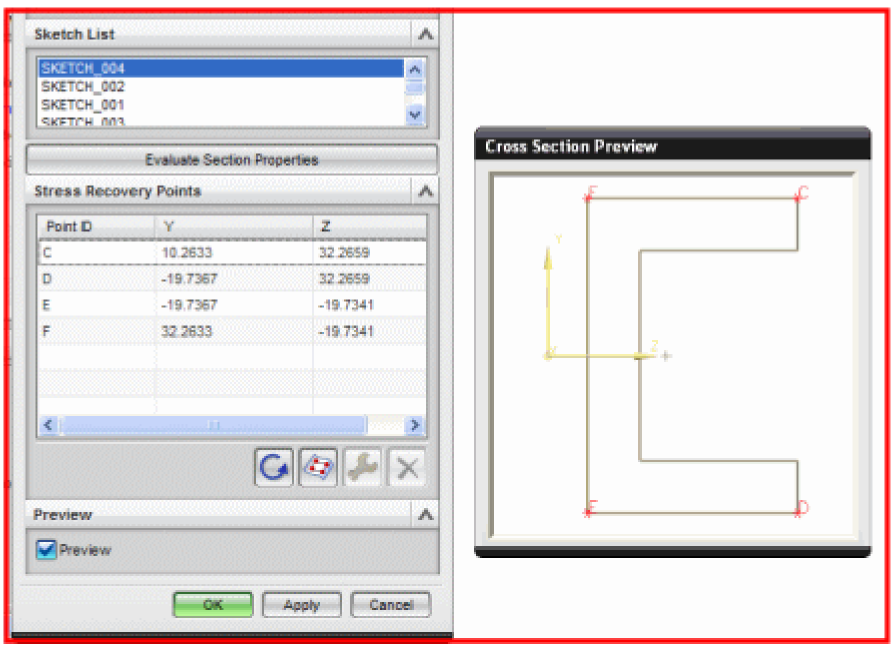
Where do I find it?

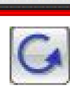
Application	Advanced Simulation
Prerequisite	An active FEM file.
Simulation Navigator	Right-click more than one mesh® **Edit Mesh Associated Data**

Default stress recovery points for beam cross sections

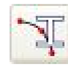
What is it?

Beginning with this release, when you create a beam cross section using the **Face of Solid** and **General Geometry** section types, the software now provides default stress recovery points.



Also, a **Reset to Defaults**  button has been added to the **Beam Section** dialog box. When you add, remove, or change the stress recovery points for a cross section, clicking this button restores the default stress recovery points calculated by the software.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation® 1D Element Section** 
Location in dialog box	**Create Section® Type**=**Face of Solid** or **General Geometry**

Meshing

Target face smoothing option for 3D swept meshes

What is it?

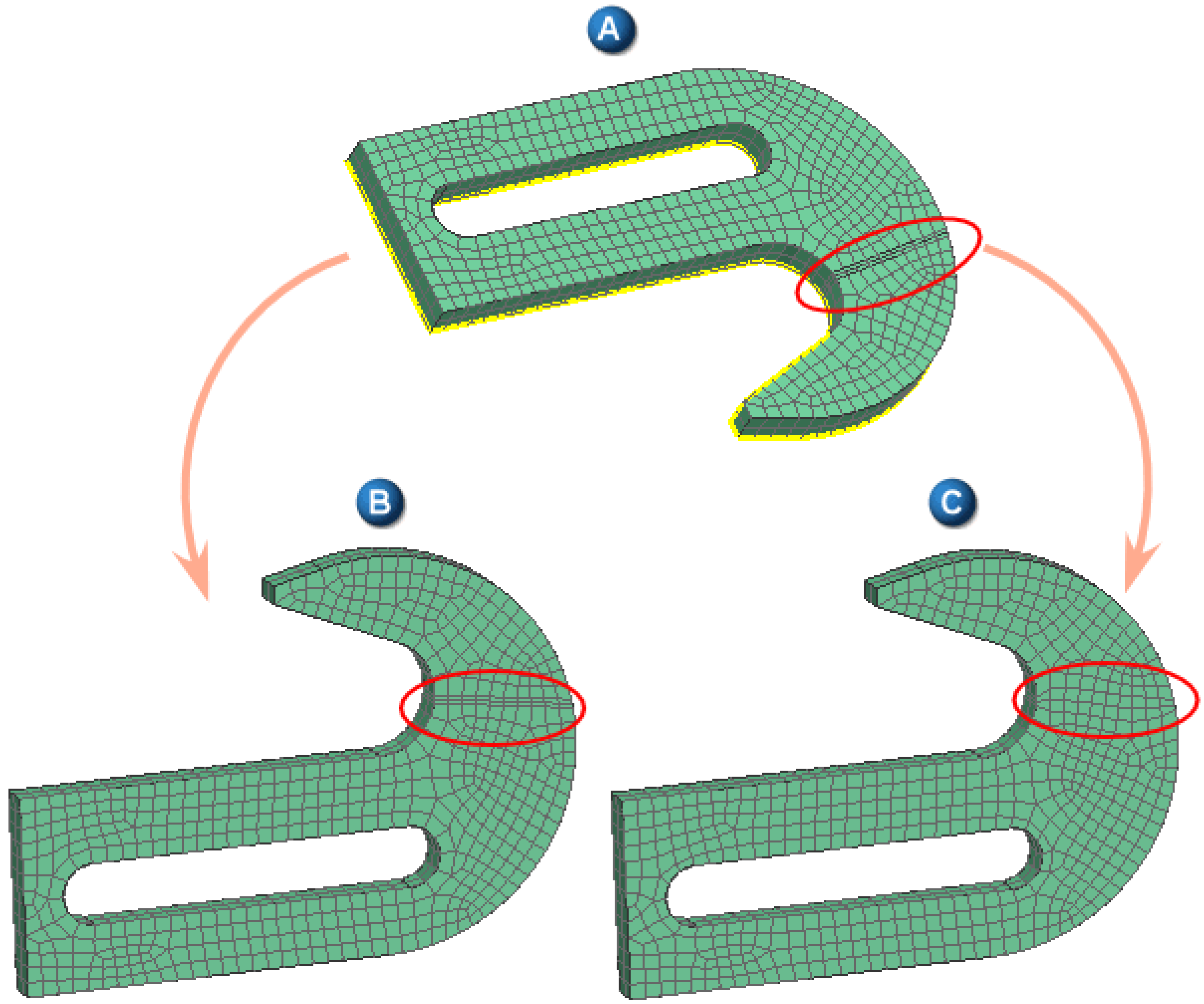
A new **Smooth Nodes** option has been added to the **3D Swept Mesh** dialog box. You can use the **Smooth Nodes** option to control whether the software smooths the nodes in the mesh on the target face. If you select the **Smooth Nodes** check box, the software makes minor modifications to the position of the nodes on the target face to achieve a more regular mesh. However, the resulting mesh on the target face will not exactly match the mesh on the source face.

The following graphic shows the effect of the **Smooth Nodes** option on the target face of a hexahedral mesh.

- (A) shows the mesh on the source face. Notice the distribution of elements in the highlighted region.

- (B) shows the mesh on the target face when the **Smooth Nodes** option is cleared. Here, the software propagates the mesh from the source to the target with no refinement on the target face. Notice that the elements in the highlighted area follow the same pattern as the elements on the source face.

- (C) shows the mesh on the target face when the **Smooth Nodes** option is selected. Here, the software refines the mesh on the target face. Notice that the elements in the highlighted area do not follow the same pattern as the elements on the source face. However, the mesh in the highlighted region now appears more regular



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 3D Swept Mesh**
Menu	**Insert® Mesh® 3D Swept Mesh**

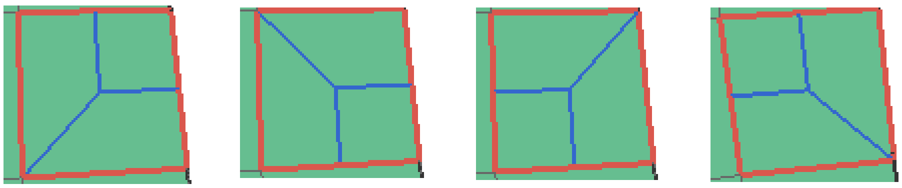
Split Shell enhancements

What is it?

This release includes several enhancements to the **Split Shell** command.

Ability to interactively define the location of the split line

Use the new **Interactive Location** option in the **Based on** list to have the software interactively preview possible split patterns as you move your mouse over the selected element. The following graphic shows an example of the four possible split patterns that the software previews for the **Quad to 4 Quads** type.



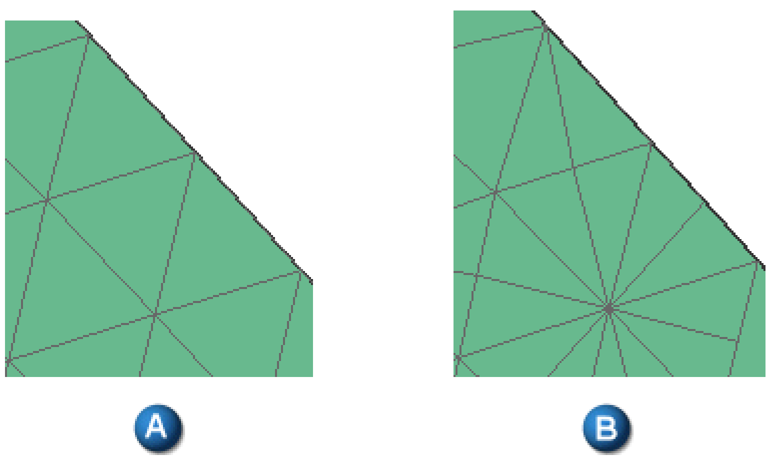
When you are satisfied with the previewed split pattern, click the mouse. The software then immediately splits the element. Because you do not have to click **OK** or **Apply** with the **Interactive Location** option, you can quickly split an element.

You can use the new **Interactive Location** method with all options on the **Type** menu except **Quad to 3 Quads** and **Triangle to 4 Triangles**.

New method for dividing triangular elements

A new **Triangles to 2 Triangles** option has been added to the **Type** list in the **Split Shell** dialog box. Use the **Triangles to 2 Triangles** option to divide a selected triangular element into two smaller triangular elements.

In the following graphic, (A) shows a small section of a mesh of triangular elements. (B) shows the same elements after they have been divided with the **Triangles to 2 Triangles** option.



Option for splitting elements based on their warp value

For the **Quad to 2 Triangles** and **Quad to 3 Triangles** types of splits, you can use the new **Element Quality** option to divide selected quadrilateral elements whose warp values exceed a specified warp threshold value. This allows you to eliminate highly warped elements from your model. How the software splits any warped elements depends on whether you want to divide the quadrilateral element into two or three triangular elements.

- With the **Quad to 2 Triangles** type, the software divides a warped element along the shorter of the element's two diagonal edges.

- With the **Quad to 3 Triangles** type, the software divides a warped element by inserting a new node in the middle of the element's longest edge.

New option to control the merging of new nodes

You can use the **Merge New Nodes** option to control whether the software merges any new nodes the software creates when it divides the elements. In previous releases, the software always merged any duplicate nodes created by the **Split Shell** command. If you clear the **Merge New Nodes** check box, the software retains all new nodes created by the **Split Shell** command.

Note If you clear the **Merge New Nodes** check box, you should use the **Nodes** option in the **Model Check** dialog box to search for duplicate nodes before you solve the model.

Where do I find it?

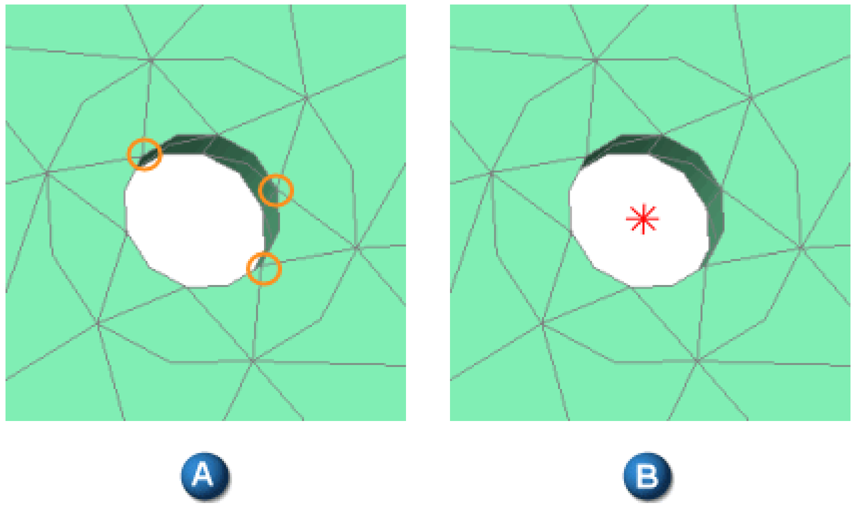
Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® Split Shell** 
Menu	**Edit® Element® Split Shell**

Ability to create a node at the center of a hole

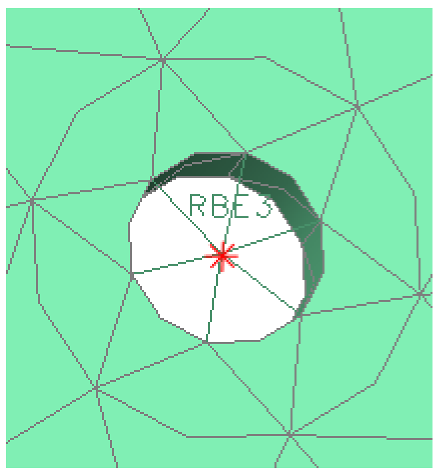
What is it?

You can now use the **Node Between Nodes** command to create a node at the center of three existing nodes. With the new **At Center of Three Nodes** option on the **Type** list, the software uses the three nodes that you select to define an arc. The software then creates the new node at the center of that arc. The **At Center of Three Nodes** option is helpful, for example, when you need to create a new node at the center of a hole in a model that does not contain any geometry.

The graphic below shows an example of the new **At Center of Three Nodes** option. (A) shows the three selected nodes. (B) shows the new node the software creates at the center of the selected nodes.



The **At Center of Three Nodes** option, is useful, for example, when you want to use a spider-type connection, such an RBE2 or RBE3 element, to connect the center of a hole to the nodes in the surrounding mesh, as shown below.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Node Operations® Node Between Nodes** 
Menu	**Insert® Node® Node Between Nodes**

Merging meshes

What is it?

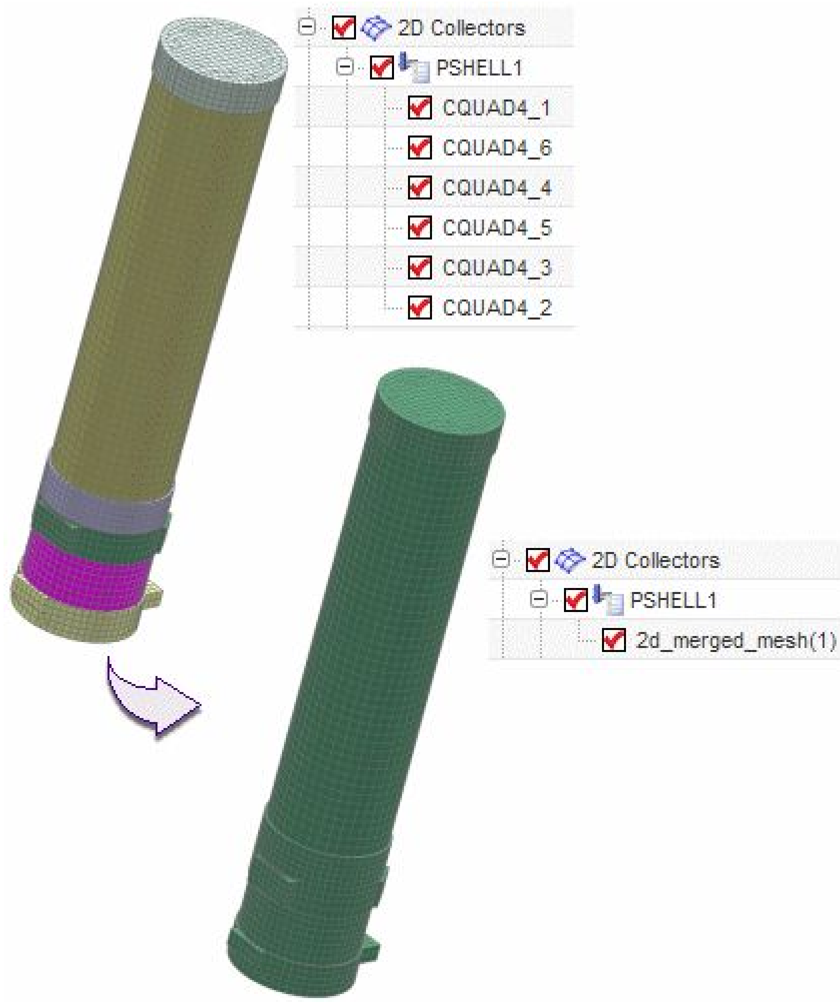
Use the new **Merge Meshes** command to merge manually created, extracted, or imported meshes in the **Simulation Navigator**. The meshes to be merged must:

- Not be associated with any geometry.
- Be located in the same mesh collector.

When you merge the meshes, the software places the meshes in a new node the **Simulation Navigator**. The software appends the string `_merged_mesh` to the name of the new mesh.

The merged mesh assumes:

- The mesh associated data of the first mesh you select.
- The mesh display properties of the parent mesh collector.


2D Collectors
PSHELL1
CQUAD4_1
CQUAD4_6
CQUAD4_4
CQUAD4_5
CQUAD4_3
CQUAD4_2
2D Collectors
PSHELL1
2d_merged_mesh(1)

Six meshes (shown with different display colors for clarity) merged into a single mesh

You can use the **Merge Meshes** command to organize your meshes in the **Simulation Navigator**. **Merge Meshes** is particularly useful when you are working with large, imported models. You can also use the **Merge Meshes** command in conjunction with the **Extract Elements** command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Meshes that are not associated with geometry and located in the same mesh collector.
Simulation Navigator	Select multiple meshes® right-click a selected mesh® **Merge Meshes**

Post-processing

Enhanced coordinate system support

What is it?

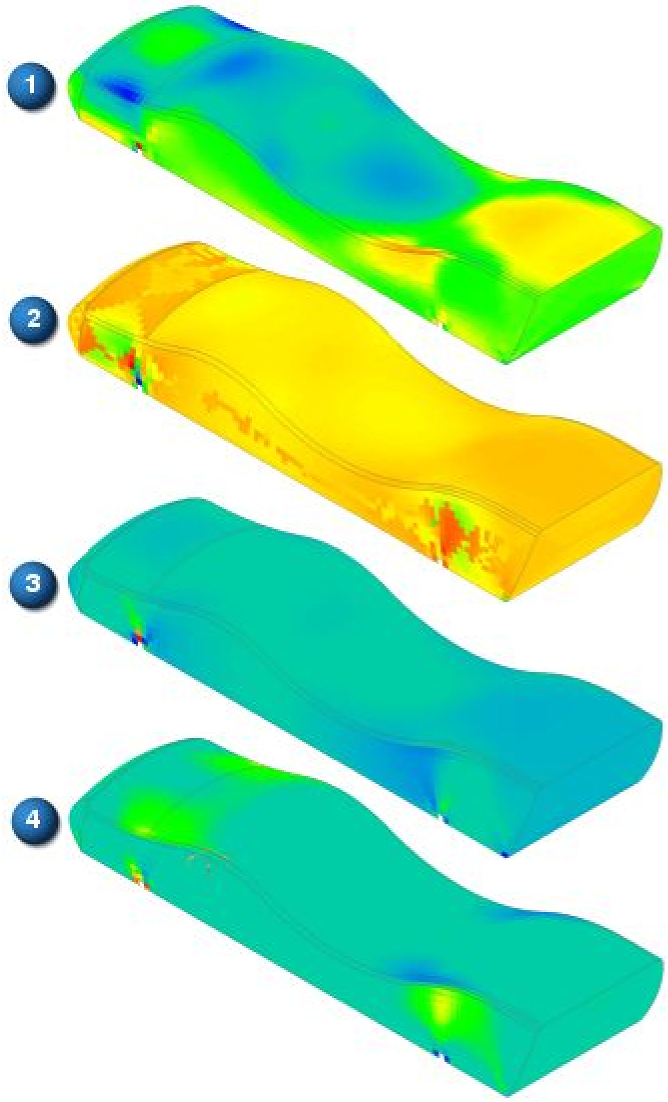
In previous releases, you could display results with respect to the absolute and work coordinate systems, using rectangular (Cartesian), cylindrical, or spherical coordinates. This release adds support for the following coordinate systems:

- **Native**
- **Material**
- **Selected Rectangular**
- **Selected Cylindrical**
- **Selected Spherical**

Coordinate system support is available in the following post-processing dialog boxes:

- **Set Result**
- **Cutting Plane**
- **Free Body Results**

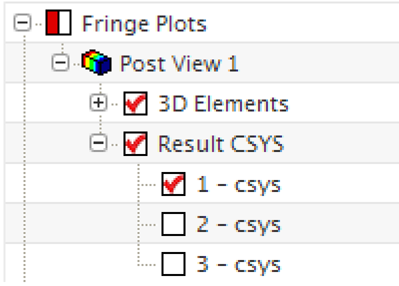
The following image compares contour plots of corresponding stress components as transformed into different coordinate systems.



(1) XY shear stress, absolute rectangular CSYS; (2) XY shear stress, untransformed native results (element CSYS); (3) 12 shear stress, material CSYS; (4) rq shear stress; selected cylindrical CSYS.

Coordinate systems in the Post Processing Navigator

Coordinate systems in the solver output file are listed under the **Post View** node in the **Post Processing Navigator**. Result coordinate systems are hidden by default.



Coordinate systems written to the solver output file include, for example, nodal coordinate systems and local coordinate systems used to define component force loads or user-defined constraints.

Selected coordinate system support

In previous releases, in order to display results in an arbitrary coordinate system, you needed to align the work coordinate system (WCS) to the target coordinate and then set the post-processing coordinate system to **Work**. In this release, you can select any coordinate system that is visible and selectable in the graphics region, or listed in the **Simulation Navigator**, to arbitrarily transform results. You can select:

- Coordinate systems listed in the solver output file.

  Expand the **Post View** node and select the visibility check box for the **Result CSYS** node to display result coordinate systems and make them selectable.

  For more information, see Transform results in a selected solver output coordinate system.

- Coordinate systems in the displayed part file (typically, the Simulation file).

  You must ensure that the displayed part coordinate systems are visible in the graphics region. To do this, you can create a multiple viewport layout, or you can control the visibility of layers containing coordinate systems.

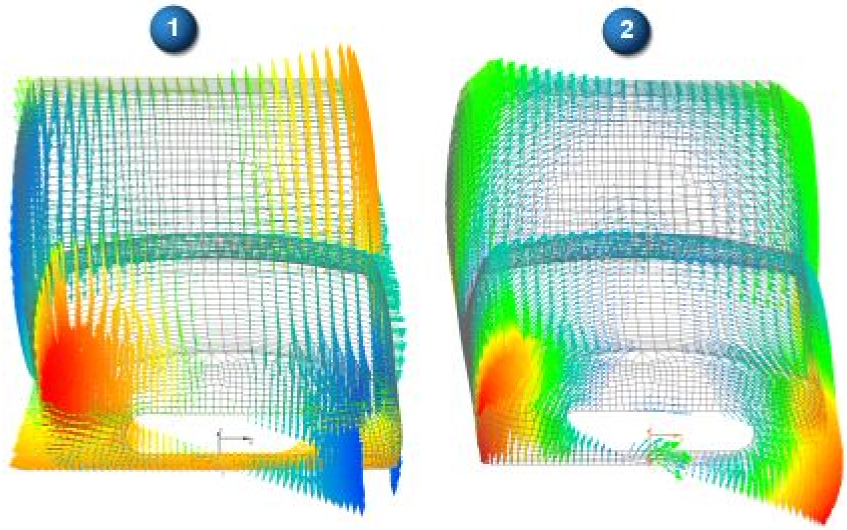
  For more information, see Graphically select a model coordinate system.

- Coordinate systems listed in the **Simulation Navigator**.

  By default, all local coordinate systems defined in the currently displayed Simulation or FEM file are listed in the **Simulation Navigator**. You can click to highlight the CSYS node in the **Simulation Navigator** to select a coordinate system when setting the result display in a post view.

  For more information see Select a coordinate system listed in the Simulation Navigator.

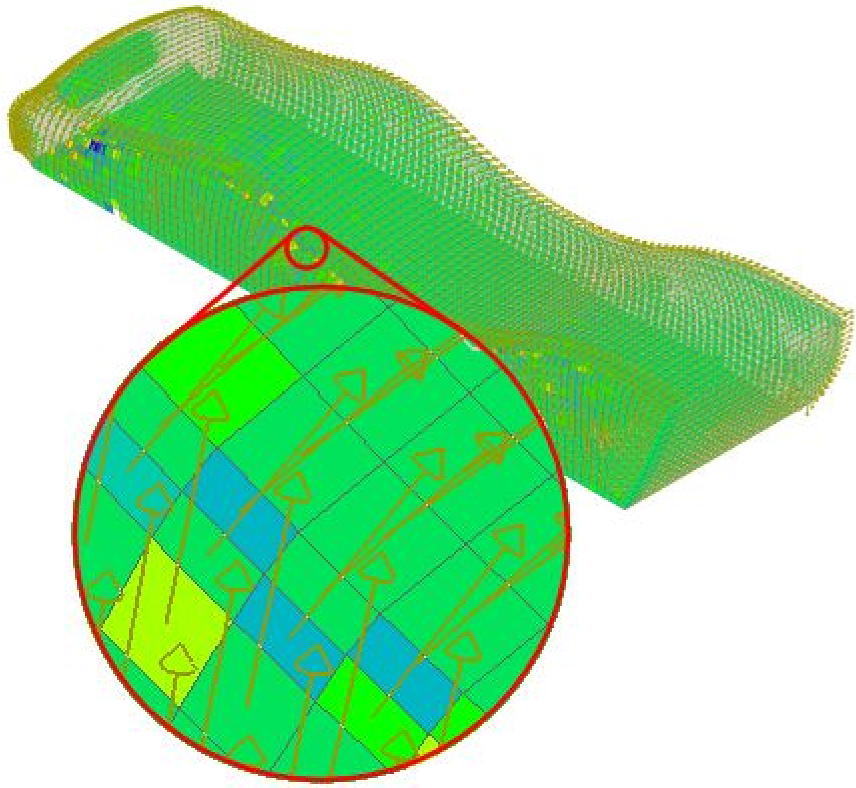
The following figure compares arrow plots of (1) XYZ displacement in the absolute rectangular coordinate system and (2) q displacement in a selected cylindrical coordinate system, for a model undergoing twisting. The selectable result coordinate system is just visible at the bottom center of each image.



Material coordinate system support

If any meshes or elements in your model have a defined material orientation (as for orthotropic or anisotropic materials or composite laminates), you can view tensor component results in the material coordinate system.

Optionally, when **Material** is selected for the coordinate system on the **Set Result** dialog box, you can click **Display Material Orientation** to temporarily display the material orientation vectors.



Native coordinate system support

Native replaces the **Local** option used in previous releases of NX. The native coordinate system is the untransformed coordinate system in which results are written to the solver output file. Typically, this is the element coordinate system, but may be some other coordinate system based on the solver, analysis type, or element formulation.

Why should I use it?

The solver writes out results as a set of component values at nodes and/or elements with respect to a coordinate system determined by the solver and element formulation. When you load results into NX, this unprocessed data is transformed into a consistent coordinate system, and derived results are calculated.

When you define the results display for a post view, you can specify the coordinate system into which component results are transformed.

When boundary conditions are defined with respect to local coordinate systems, transforming results into those coordinate systems can produce cleaner and more meaningful results displays.

Where do I find it?

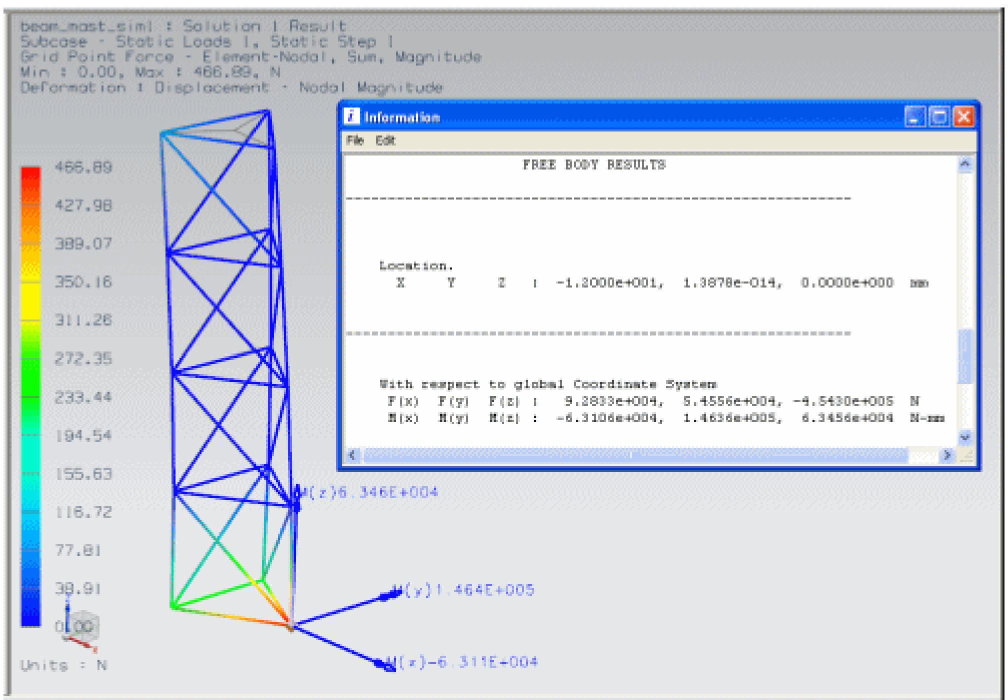
Application	Advanced Simulation
Prerequisite	To view results in the material coordinate system, your model must define material orientation. To view results in a selected coordinate system, the solver output file or the displayed part must include defined coordinate systems.
Toolbar	**Post Processing** toolbar ® **Set Result**
Menu	**Tools® Result® Set Result**

Free body results display

What is it?

This release introduces the **Free Body Results** command. Use this command to display the total forces and moments for a specified interface summed about a specified node or a point in space. The interface is defined by nodes connected to a group of elements. You can display the results with respect to the global coordinate system or another selected coordinate system.

The command lists the forces and moments in the **Information** window and displays the magnitude or component vectors in the graphics window.



This command is supported only for solvers that can output grid point force and moment results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved solution containing **Grid Point Force** results, and a post view displayed.
Post Processing Navigator	Right-click the **Post View** node® **Free Body Results**

Sum option for grid point force and moment element-nodal results

What is it?

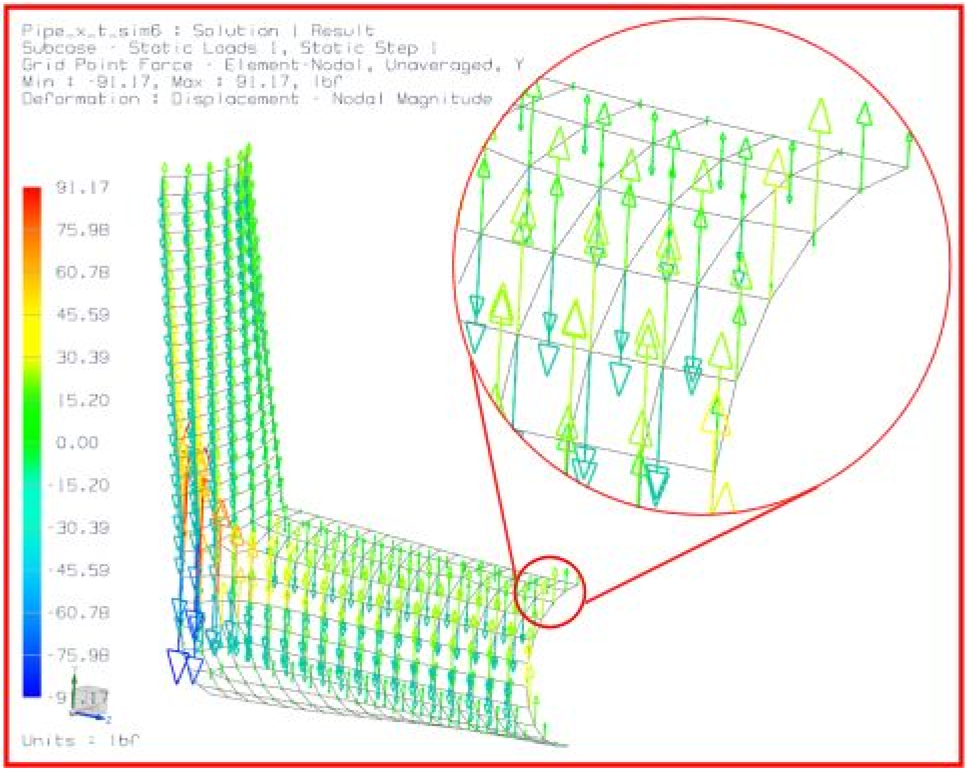
You can now display force or moment results summed at the nodes of all elements or a subset of the elements in your model.

In the **Set Result** dialog box, the **Averaged** check box has been replaced with a list named **Nodal Combination**, with three options:

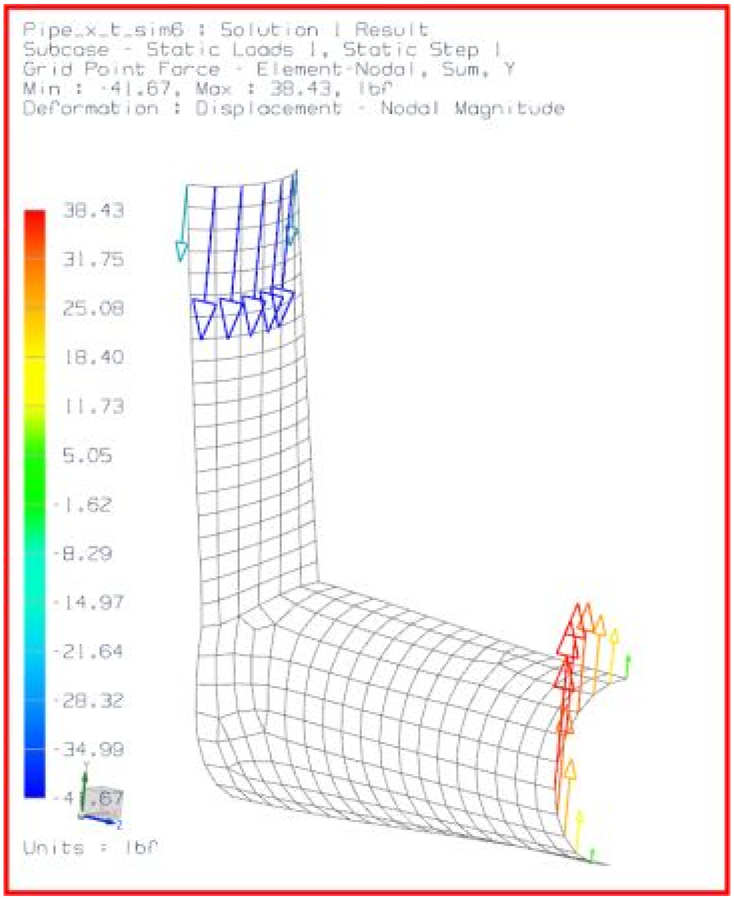
- **None**
- **Average**
- **Sum**

At the nodes of the elements in consideration, the new **Sum** option sums the force data (for grid point force results) or moment data (for grid point moment results). This option is available only for **Grid Point Force — Element-Nodal** and **Grid Point Moment — Element-Nodal** results, and only if the structural output request for your solution specifies **Grid Point Force**.

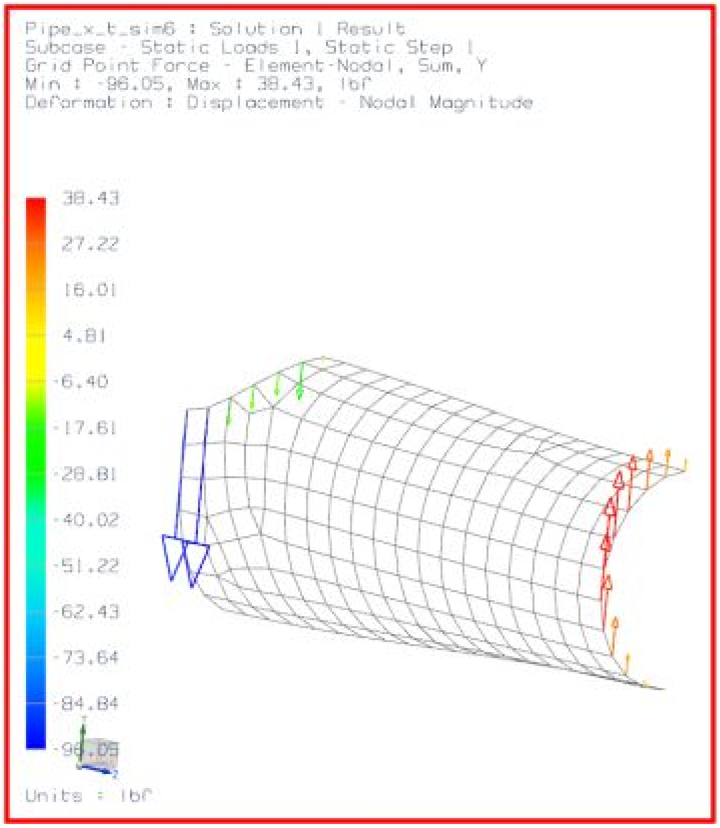
The **Exclude Elements Not Displayed** option, which is available only with the **Sum** option, lets you restrict the summation to the nodes in the elements displayed in the post view. You can limit the post view display to a group of elements by using the **Show Only** command on the group or by turning off individual meshes in the **Post Processing Navigator**.



Grid point force arrow plot; Nodal Combination = None



Nodal Combination = Sum



Nodal Combination = Sum; some elements hidden; Exclude Elements Not Displayed option selected

Where do I find it?

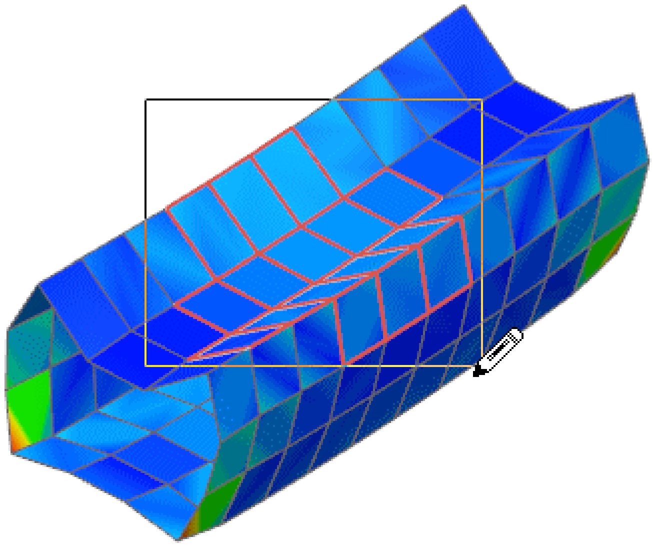
Application	Advanced Simulation
Prerequisite	A solved solution containing **Grid Point Force** results, and a post view displayed.
Post Processing Navigator	Right-click the **Post View** node® **Set Result**
Location in dialog box	**Nodal Combination** list® **Sum**

Box selection methods for elements and nodes in post-processing

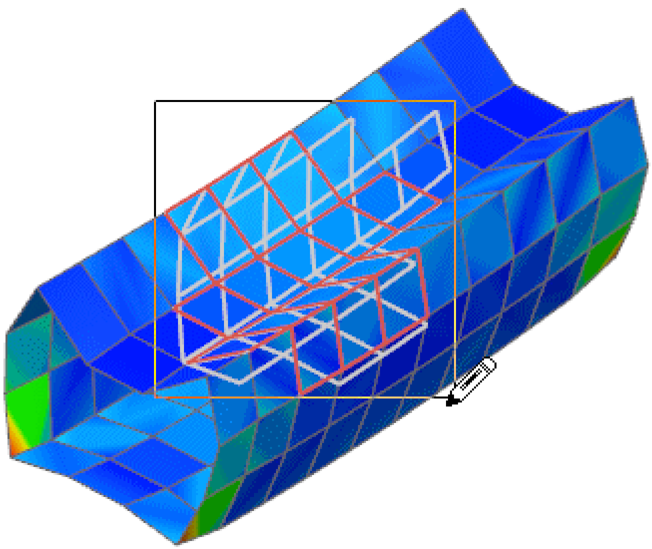
What is it?

This release introduces two new methods for selecting nodes and elements in the **Identify** command and the new **Free Body Results** command dialog boxes: **Box (Visible)** and **Box (All)**.

- **Box (Visible)** — Lets you drag a selection box to select nodes or elements that are visible in the model's current orientation in the graphics window. Nodes or elements that are hidden behind other post view elements are not selected.



- **Box (All)** — Lets you drag a selection box to select nodes or elements. Nodes and elements that are hidden behind other post view elements are also selected.



Where do I find it?

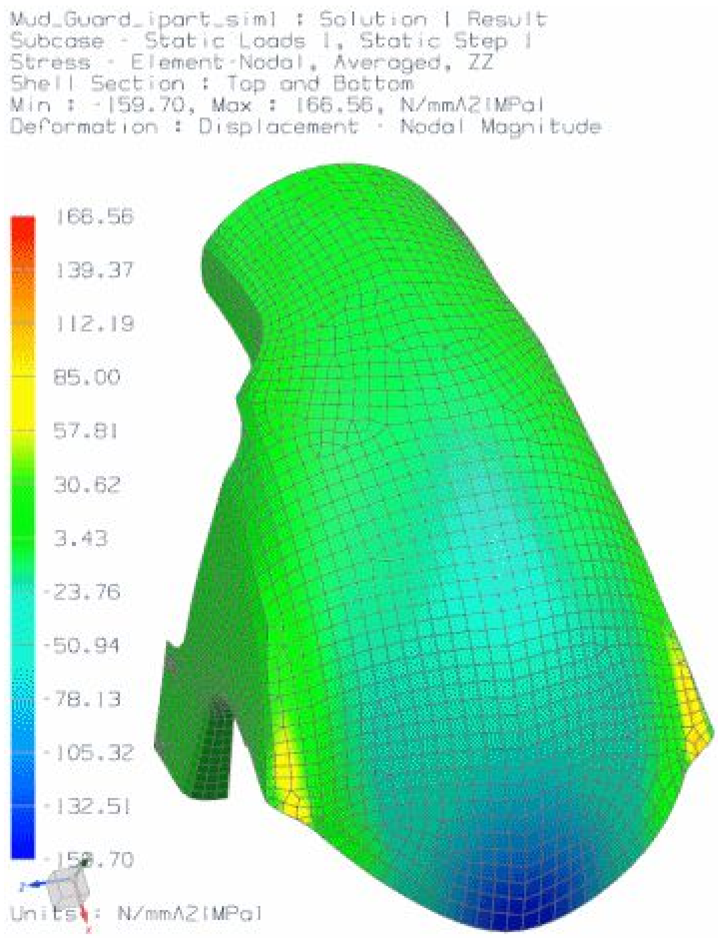
Application	Advanced Simulation, Design Simulation
Prerequisite	A solved solution and a post view displayed.
Post Processing Navigator	Right-click the **Post View** node® **Free Body Results** or **Identify**
Location in dialog box	**Pick** list® **Box (Visible)** or **Box (All)**

Top and bottom stress and strain results for 2D elements

What is it?

You can now display the stress or strain results on both the top and bottom of 2D elements simultaneously using the new **Top and Bottom** option, which has been added to the **Location: Shell** list in the **Set Result** dialog box.

This option is available for **Smooth**, **Banded**, and **Element** displays. It is not supported in **Cutting Plane** displays.



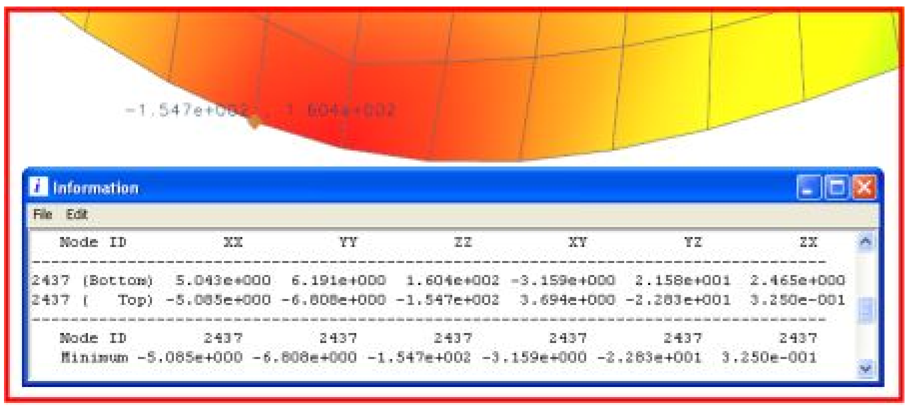
The shell top corresponds to the positive element normal. The shell bottom corresponds to the negative element normal.

Note This option uses the **Backface Culling** visualization preference to display the result correctly. When you request **Top and Bottom** stress or strain results, you are prompted to turn on **Backface Culling**. The **Backface Culling**  button is now located on the **Post Processing** toolbar. For more information, see **Unsatisfied xref title**.

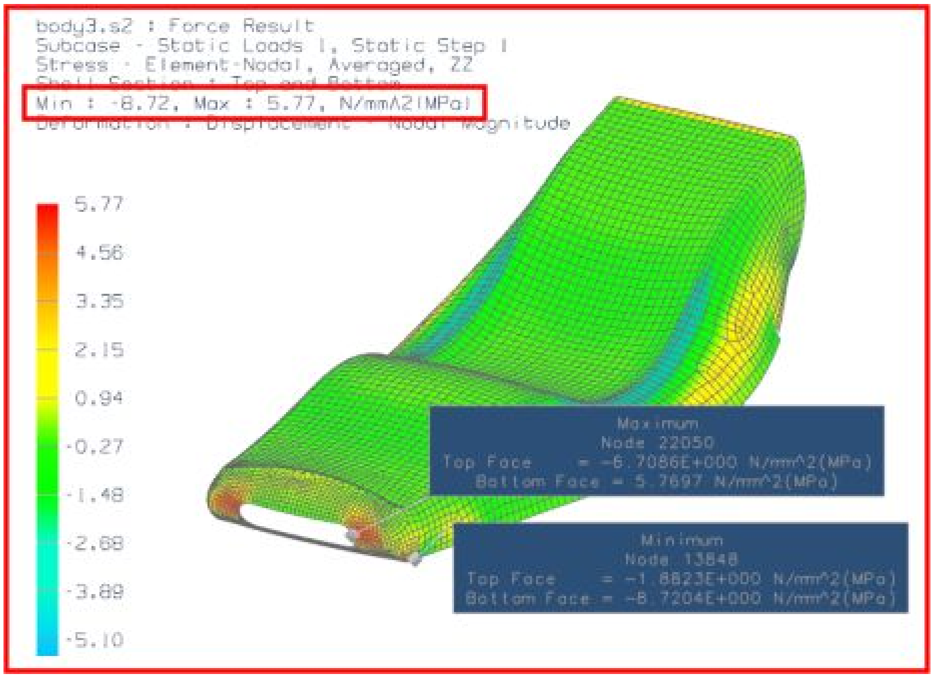
When you use the **Graph**, **Identify**, and **Marker** commands on **Top and Bottom** results, note the following:

- When you create a graph on a top-and-bottom display, the results are extracted from the top of the elements.

- When you use the **Identify** command, both top and bottom results are listed.



- When you use the **Marker** command, the marker boxes list both the top and bottom results at the minimum and maximum locations. However, the contour plot heading always displays the minimum and maximum of the entire model, regardless of the top or bottom face.



Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved solution and a **Smooth**, **Banded**, or **Element** post view displayed.
Toolbar	**Post Processing** toolbar ® **Set Result**
Location in dialog box	**Location: Shell** list® **Top and Bottom**

Bending stress and strain results for 2D elements

What is it?

You can now display bending stress or strain results for 2D shell elements using the new **Bending** option, which has been added to the **Location: Shell** list in the **Set Result** dialog box.

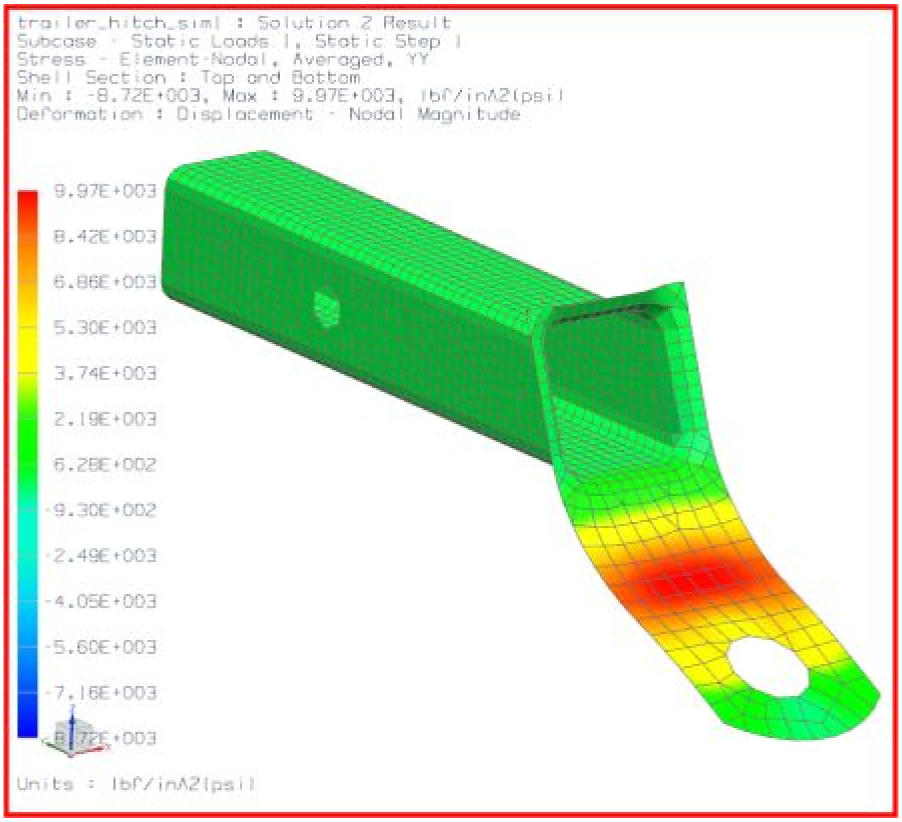
This option is available for **Smooth**, **Banded**, and **Element** contour displays. It is not supported in **Cutting Plane** displays.

The **Bending** option lets you isolate the bending stress, without including the other components (such as membrane stress and shear stress) that have the same stress value at the top and bottom of the element. The bending results are calculated from the top and bottom stress results as:

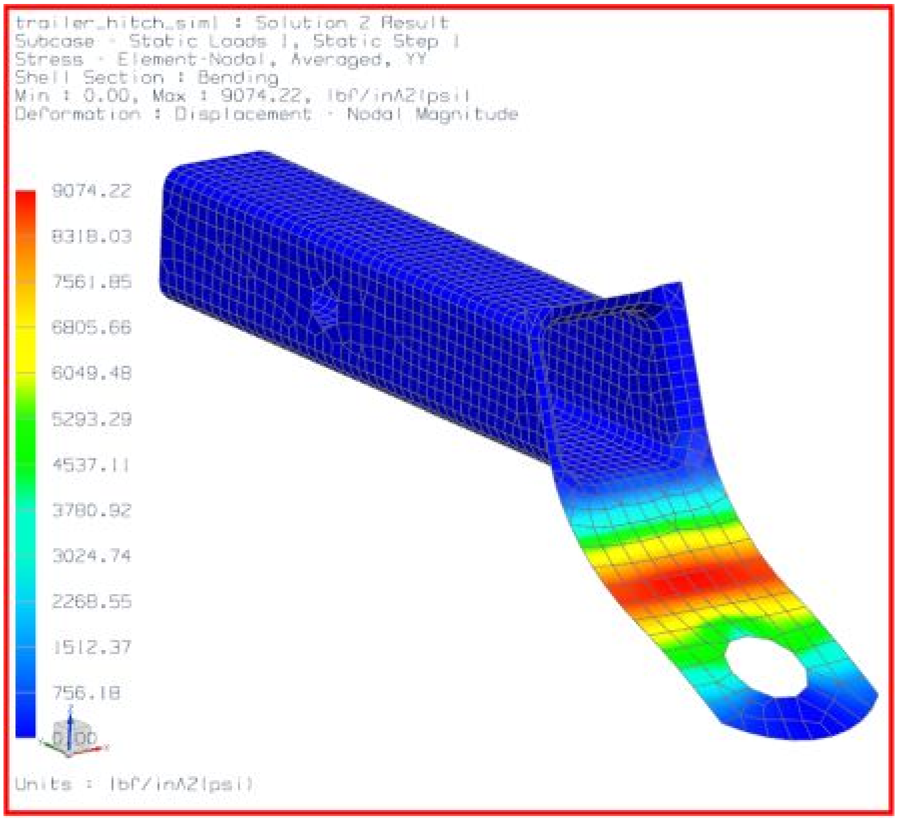
$s_B = 1/2\ (s_{TS} - s_{BS})$

where s_{TS} is the selected stress component from the top surface of the element and s_{BS} is the selected stress component from the bottom surface.

In the following example, the model is loaded in the axial and vertical directions. The difference between the top and bottom stress result and the bending stress result is relatively small, which indicates that most of the stress is due to bending.



Top and Bottom stress result, 9970 psi



Bending stress result, 9074.22 psi

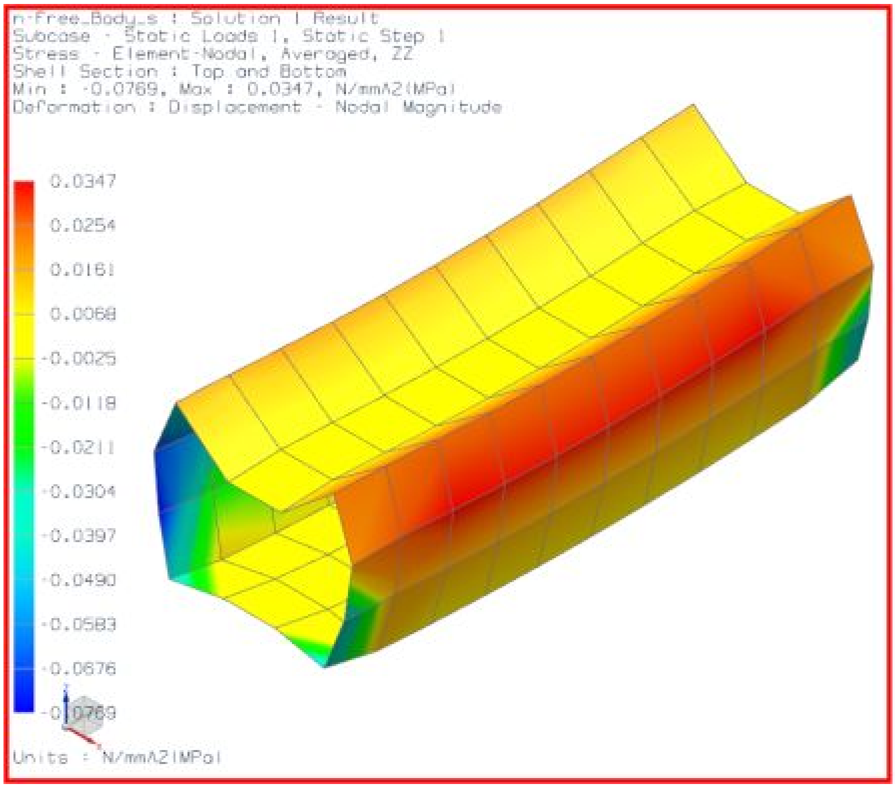
Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved solution and a **Smooth**, **Banded**, or **Element** post view displayed.
Toolbar	**Post Processing** toolbar ® **Set Result**
Location in dialog box	**Location: Shell** list® **Bending**

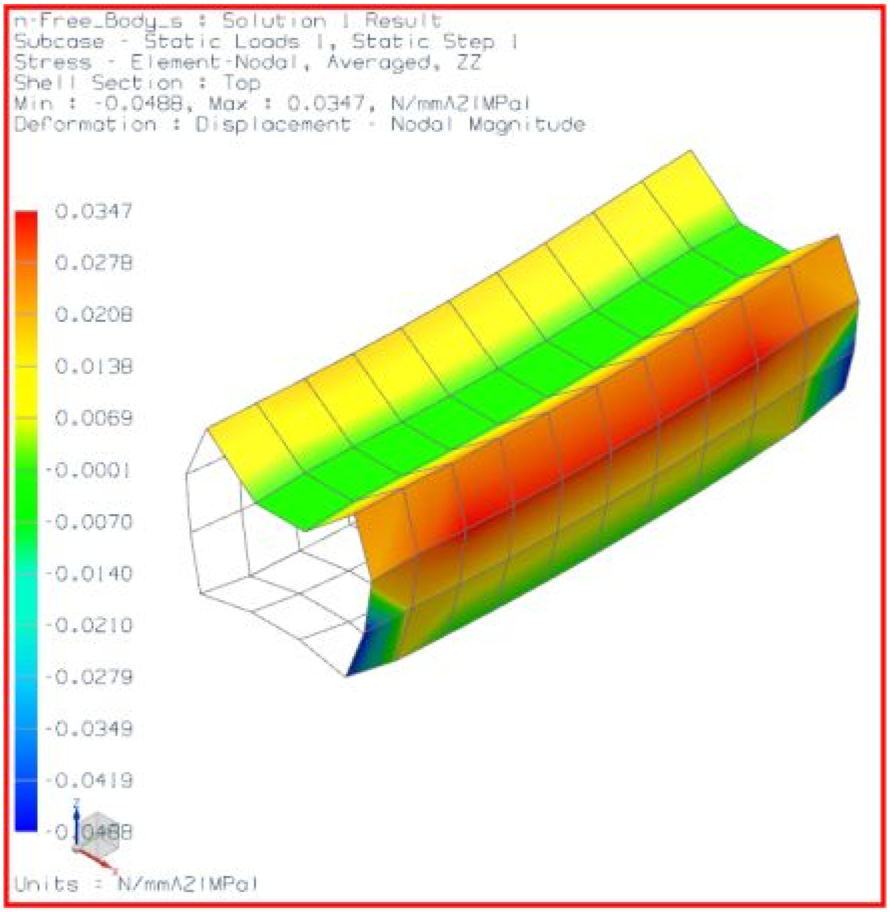
Backface Culling preference added to Post Processing toolbar

What is it?

The **Backface Culling** visualization preference has been added to the **Post-Processing** toolbar for your convenience. The primary use for this preference is to simultaneously display stress or strain results on both the top and bottom of 2D elements. However, it is also useful for improving graphics performance, and for quickly determining whether 2D element normals are consistent across your model (the element is shaded only on the positive normal side).



Shell top and bottom stress results with Backface Culling turned on



Shell top stress results with Backface Culling turned on

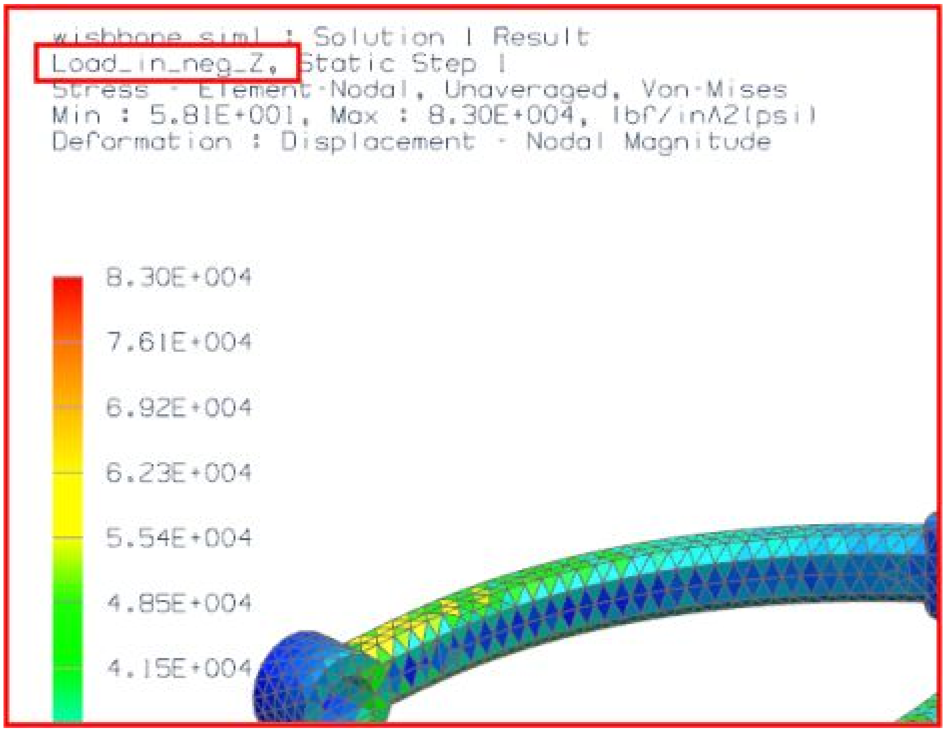
Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved solution and a post view displayed.
Toolbar	**Post Processing** toolbar ® **Backface Culling** 

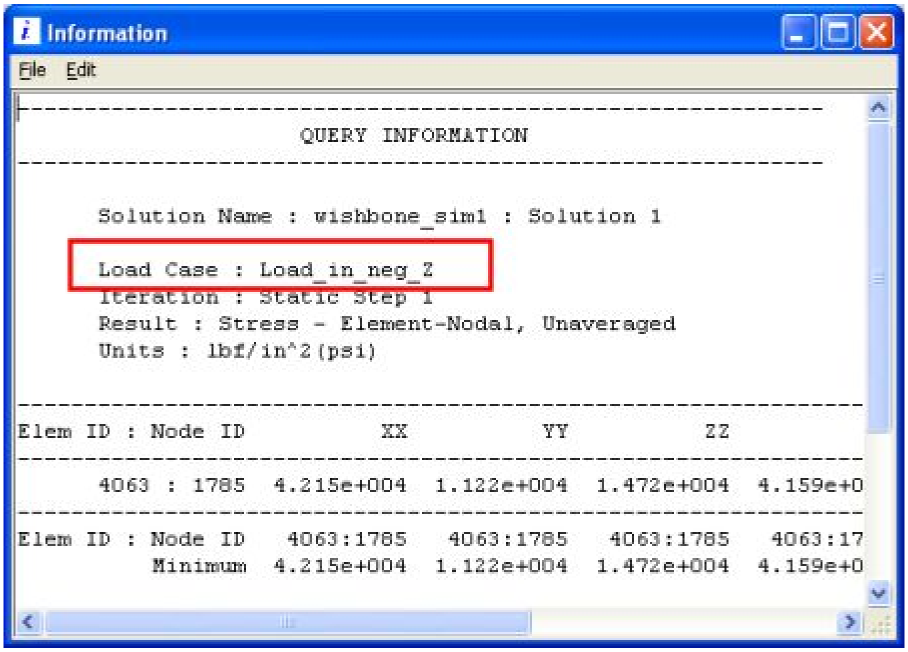
Load case names in post-processing

What is it?

Post views and results windows in post-processing now display the load case names defined in Advanced Simulation or in the solver result file (for imported results). In previous releases, load case names in post-processing were generated dynamically when you displayed the results.



“Load_in_neg_Z” was defined as the solution step subcase name




```
---------------------------------------------------------------
                     QUERY INFORMATION
---------------------------------------------------------------

      Solution Name : wishbone_sim1 : Solution 1

      Load Case : Load_in_neg_Z
      Iteration : Static Step 1
      Result : Stress - Element-Nodal, Unaveraged
      Units : lbf/in^2(psi)

------------------------------------------------------------------
Elem ID : Node ID           XX           YY           ZZ
------------------------------------------------------------------
      4063 : 1785  4.215e+004  1.122e+004  1.472e+004  4.159e+0
------------------------------------------------------------------
Elem ID : Node ID   4063:1785   4063:1785   4063:1785   4063:17
          Minimum  4.215e+004  1.122e+004  1.472e+004  4.159e+0
```

Load case name displayed in Information window from the Identify command

Import LS-Dyna results

What is it?

You can now import d3plot result files from the LS-DYNA solver.

The **Import Results File** command includes a new **Files of Type** filter named **LS-DYNA State Databases** that allows you to select the d3plot results file. When you import this file, all associated d3plot*n* children files are imported automatically.

Post-processing of transient steps is supported.

Where do I find it?

Application	Advanced Simulation
Post Processing Navigator	Right-click the **Imported Results** node® **Import Results→Browse** button
Location in dialog box	**Files of Type** list® **LS-DYNA State Databases(*.*)**

Nastran support enhancements

Changes to parameter specifications

What is it?

This release includes several enhancements to the support for the Nastran PARAM bulk data entry in Advanced Simulation.

Ability to specify solution parameters on export from a FEM file

When you have a FEM file active, the **Export Simulation** dialog box now includes a new **Additional PARAMs** option in the **Output Options** group. This allows you to include an existing **Solution Parameters** modeling object in the Nastran input file.

Customer defaults support for parameters

This release includes a new **Additional Parameters** option in the customer defaults that you can use to specify a list of parameters that you want the software to include in every Nastran input file you export or solve. The software writes out the PARAM options and values that you specify each time you export or solve a solution.

Additionally, you can use the **Additional Parameters** option to override any parameters that are hardcoded at export by the software, such as PARAM,POST,-2 or PARAM,OMACHPR,YES.

To access this option:

1. From the **File** menu, choose **Utilities→Customer Defaults**.

2. In the **Customer Defaults** dialog box, choose **Simulation→Extras**.

3. On the **Nastran** tab, use the **Additional Parameters** section to enter the parameters you want the software to write out each time you export or solve a solution.

   - You must include the `PARAM` keyword.

   - If you specify multiple parameters, you must separate them with commas.

Solution Parameters modeling object updated

The **Solution Parameters** modeling object has been updated to include all supported parameters for the NX Nastran 7.0 and MSC Nastran v. 2008 releases.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver

Automatic detection of units on import

What is it?

When you import a Nastran input file (.dat or .op2) into NX, you can have the software use the units specified in the file as the unit system for the model. When you select the new **User solver file units if available** option in the **Import Simulation** dialog box, the software searches for the `PARAM,UNITSYS` bulk data entry in the unit file.

- If the `PARAM,UNITSYS` entry is present, the software imports the file in the unit system specified by `PARAM,UNITSYS`.

- If the `PARAM,UNITSYS` entry is not present, the software imports the file in the unit system you specified in the **Import Simulation** dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Import® Simulation**

Option to export DOF Sets in the alternate format

What is it?

When you export a model that contains **DOF Sets**, you can now choose to export those sets using the appropriate alternate format bulk data entries. If you select the new **Write DOF Sets Using Alternate Format** option on the **Formatting Options** tab in the **Solver Parameters** dialog box, the software writes out any **DOF Sets** in their alternate format. For example, the software writes out analysis degree-of-freedom sets as ASET1 bulk data entries rather than ASET entries.

Most alternate format degree-of-freedom entries use the THRU keyword to specify ranges of nodes. In contrast, the standard degree-of-freedom set entries require a separate entry for each node. Consequently, using the alternate format reduces the size of the Nastran input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Simulation Navigator	Right-click an existing Simulation® **Edit Solver Parameters**

Changing the orientation of a model on export

What is it?

You can now export a solver input file in a coordinate system that you specify. In the **Export Simulation** dialog box, you can use the new **Model Orientation** options to export the model in a different coordinate system than the one used in the original Simulation file. This allows you to re-orient the model on export to a specified coordinate system.

If you select a different coordinate system on export, the software replaces the basic coordinate system references in your Nastran input file with a Cartesian coordinate system that is equivalent to the one you select. The software updates all entities in your model, such as nodes, loads, or material orientation vectors, that referred to the basic coordinate system, to refer to the new coordinate system. This changes the orientation of the model.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Export® Simulation**

Coordinate system names retained on export and import

What is it?

When you export a model, NX now uses comment ($) bulk data entries to preserve the names of any coordinate systems in your model. The software writes out the name of the coordinate system immediately before the associated CORDx bulk data entry in your input file. For example:

```
$*
$* COORDINATE SYSTEM CARDS
$*
$*  NX Coordinate System: 100 - csys::w7_f
CORD2R*              100              05.7600000000E+020.0000000000E+00+
*       0.0000000000E+005.7600000000E+020.0000000000E+001.0000000000E+00+
*       5.7670710678E+027.0710678119E-010.0000000000E+00                +
*
ENDDATA
```

If you later re-import an ASCII format .dat input file back into NX, the software preserves any coordinate system names. Comment ($) bulk data entries are not stored in the binary format .op2 file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Export® Simulation** **File® Import® Simulation**

Ability to specify subcase labels for solution steps

What is it?

Use the new **Use Step Name as Label** option in the **Solution Step** dialog box to control whether the software uses the specified **Name** for the solution step as the label for the corresponding Nastran subcase. When you export or solve your model, the software uses the specified **Name** to write out a `LABEL` case control command for the associated `SUBCASE` case control command in the Nastran input file.

In previous releases, you had to manually specify a label for each solution step. Beginning in this release, NX searches imported OP2 files for subcase labels. It then displays the subcase labels in the **Post-Processing Navigator**. To facilitate results management in the **Post-Processing Navigator**, you should associate a meaningful label with each subcase in your solution.

This release also includes a new **Customer Default** that you can use to control whether the **Use Step Name as Label** option is selected by default. To access this option:

1. From the **File** menu, choose **Utilities→Customer Defaults**.

2. In the **Customer Defaults** dialog box, choose **Simulation→Extras**.

3. On the **Nastran** tab, either select or clear the **Use Step Name as Label** option.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Simulation Navigator	Right-click an existing solution and select **New Subcase**

Export nodes as fluid grid points

What is it?

Currently, you cannot directly designate nodes as fluid grid points for coupled fluid-structural analyses in Advanced Simulation. However, when you export or solve your model, the software now checks each node to see whether it belongs to a solid element that is assigned a fluid material (`MAT10` bulk data entry). If element's PSOLID physical property references a fluid material, NX writes out a value of -1 for CD in field 7 of the corresponding `GRID` bulk data entry. This designates the node as a fluid grid point.

Note Note that the existing displacement coordinate system in field CD on the GRID card must be the BASIC (0) coordinate system for the exporter to replace it by -1. Otherwise, the software issues an error.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Export® Simulation**

Nastran import and export support enhancements

What is it?

This release includes improved import and export support for Nastran bulk data entries and case control commands.

Name	NX Nastran import/export support	MSC Nastran import/export support	Newly supported fields and other enhancements	Notes
ASET1	Yes	Yes	In previous releases, ASET1 entries were always exported as ASET entries. They can now be exported as ASET1 entries.	**Unsatisfied xref title**
BNDFIX1	Yes	Yes	• BNDFIX1 entries are now imported as a DOF set and a constraint. • In previous releases, BNDFIX1 entries were always exported as BNDFIX entries. They can now be exported as BNDFIX1 entries	**Unsatisfied xref title**
BNDFREE1	Yes	Yes	• BNDFREE1 entries are now imported as a DOF set and a constraint. • In previous releases, BNDFREE1 entries were always exported as BNDFREE entries. They can now be exported as BNDFREE1 entries	**Unsatisfied xref title**

BSET1	Yes	Yes	In previous releases, BSET1 entries were always exported as BSET entries. They can now be exported as BSET1 entries.	**Unsatisfied xref title**
CSET1	Yes	Yes	In previous releases, CSET1 entries were always exported as CSET entries. They can now be exported as CSET1 entries.	**Unsatisfied xref title**
NLPARM case control command	Yes	Yes	In previous releases, a NLPARM command for SUBCASE 1 was referenced at the Solution level. Therefore, you couldn't change the settings of NLPARM and NLPCI from one solution step to another. Now, NX reads the NLPARM command for each subcase. This means that NLPARM is referenced in the appropriate solution steps (subcases) in NX.	
OMIT1	Yes	Yes	In previous releases, OMIT1 entries were always exported as OMIT entries. They can now be exported as OMIT1 entries.	**Unsatisfied xref title**
RIGID case control command	Yes	Yes	The RIGID case control command is now supported for import and export with MSC Nastran.	
RLOAD2	Yes	Yes	The RLOAD2 entry is converted to an RLOAD1 entry on import. On export, the software exports the RLOAD1 entry.	
SUPORT1	Yes	Yes	In previous releases, SUPORT1 entries were imported as SUPORT entries. Now, SUPORT1 entries are imported as SUPORT1 entries.	

USET1	Yes	Yes	In previous releases, USET1 entries were imported as USET entries. Now, USET1 entries are imported as USET1 entries.	

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Import® Simulation** **File® Export® Simulation**

Abaqus support enhancements

Reference nodes for bolts now imported

What is it?

When you import an Abaqus input file that contains a pre-loaded bolt modeled with beam or continuum (solid) elements, NX now imports the `NODE` parameter for the `*PRE-TENSION SECTION` keyword. In Abaqus, you use the `NODE` parameter to specify the reference node across which the software applies the bolt pre-load to the pre-tension section.

In previous releases, NX did not import the `NODE` parameter for the `*PRE-TENSION SECTION` keyword. When you exported a model that contained a **Bolt Pre-Load** constraint, NX automatically assigned a reference node to the associated `*PRE-TENSION SECTION` keyword. Now, if you import an Abaqus input file that contains an existing reference node for a pre-tension section, NX uses that reference node when it creates the corresponding **Bolt Pre-Load** constraint in your Simulation file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file displayed with Abaqus as the specified solver
Menu	**File® Import® Simulation**

ANSYS support enhancements

Including comments or files in an ANSYS input file

What is it?

You can now include specific text in an ANSYS input file. Use the new **User Defined Text** modeling object to specify either text or an entire file to add either to a particular solution or within a particular step. If you choose to include a file, the software uses the ANSYS /INPUT command to insert the file.

- If you include a **User Defined Text** object in a **Solution**, you can control whether the software inserts the text:
  - At the beginning of the ANSYS input file (after the /PREP7 command).
  - At the end of the element block (after the EBLOCK command).
- If you include a **User Defined Text** object in a **Solution Step**, you can control whether the software inserts the text:
  - At the beginning of the step (before any of the cumulative option commands such as DCUM or FCUM).
  - At the end of the step (before the LSWRITE command).

For example, you can use the **User Defined Text** modeling object to insert an external file that contains a portion of your ANSYS input file. The file can contain, for example, model definition data, comment lines, or other references to external files.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation with ANSYS as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects** 
Menu	**Insert® Modeling Objects**

Contact now supported in modal and buckling solutions

What is it?

You can now use the **Surface-to-Surface Contact** command in **Modal** or **Buckling** type solutions. In previous releases, the **Surface-to-Surface Contact** command was only available in **Linear Statics** and **Nonlinear Statics** type solutions.

You can use the **Surface-to-Surface Contact** command to model the interaction between contacting surfaces. For example, you can use the **Surface-to-Surface Contact** command to include the effects of friction in a modal analysis.

Where do I find it?

Application	Advanced Simulation
Prerequisites	An active Simulation file with ANSYS as the specified solver and either **Model** or **Buckling** as the specified **Solution Type**.
Toolbar	**Advanced Simulation→Surface-to-Surface Contact** 

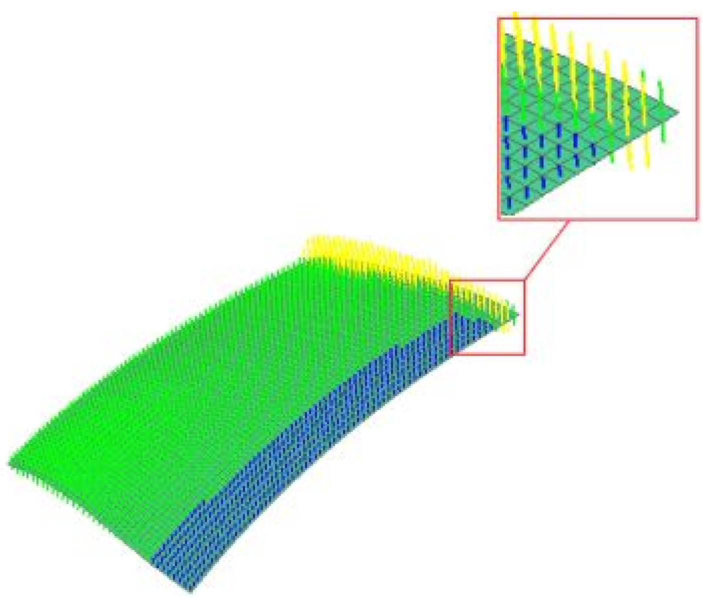
LS-DYNA support enhancements

Improvements in display capabilities

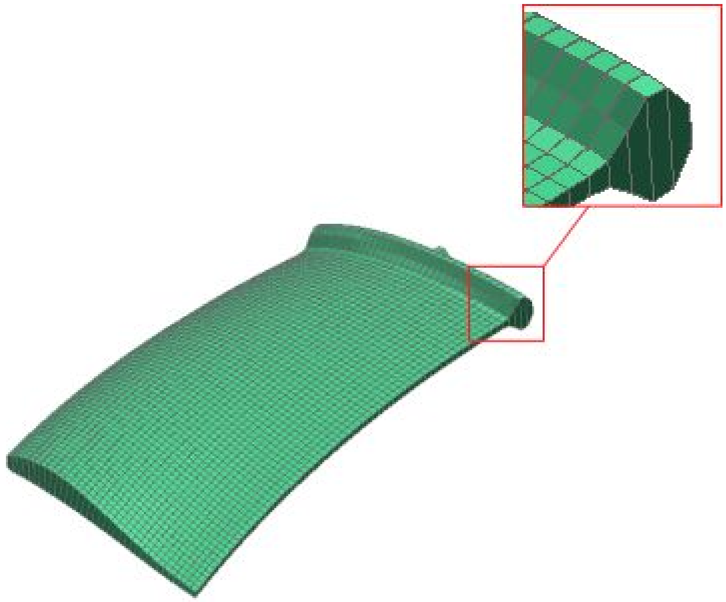
What is it?

When you are working in the LS-DYNA environment, you can now create several types of pre-processing displays to help validate your model.

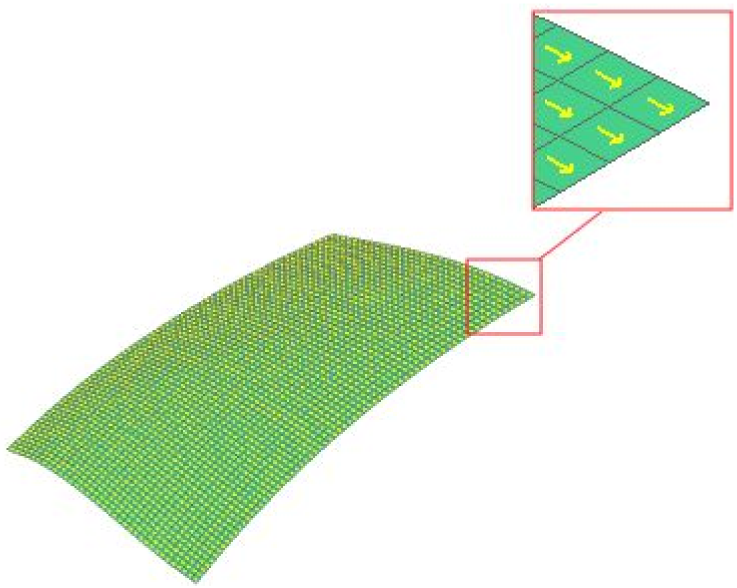
- Use the **Thickness Information** command in the Simulation Navigator to create a “hedgehog” type display of the 2D element thickness values in your model. With the hedgehog display, the software uses both colors and the relative length of lines (drawn at the centroid of each element) to indicate the variance in thickness values across the mesh. This type of thickness display is temporary and only lasts until you refresh your graphics window.



- Use the **Display 2D Element Thickness and Offset** option in the **Mesh Display** dialog box to display the elements with their assigned thickness and offset values. In this type of display, the shell elements look like solid elements.



- Use the **Element Material Orientation** options in the **Model Check** dialog box to view the direction of the assigned material coordinate system. For shell elements, an arrow displays on each element to indicate the X-axis of the material coordinate system. For solid elements, you can view one, two, or three directions of the material coordinate systems.



In previous releases, these types of displays were not supported for models in the LS-DYNA environment.

Where do I find it?

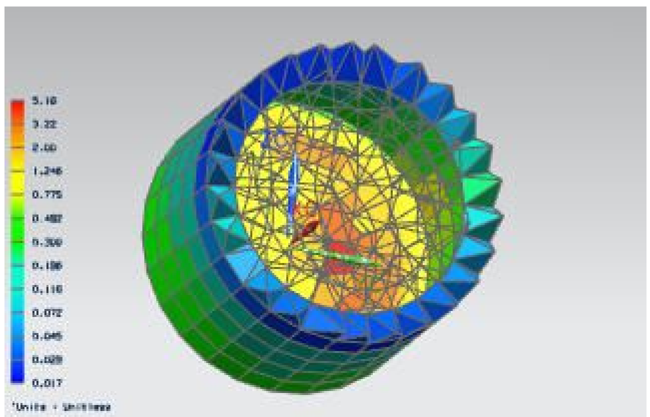
Application	Advanced Simulation
Prerequisite	An active FEM with LS-DYNA as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects**
Menu	**Insert® Modeling Objects**

Durability

Pyramid element support

What is it?

Durability results can now be calculated and displayed on pyramid elements, including the NX Nastran CPYRAM elements.



Fatigue safety factor results

Simulation Navigator enhancements

What is it?

The **Simulation Navigator** has the following enhancements in the Durability solution process.

The **Status** column for a Durability results node displays **Obsolete** when you:

- Deactivate an event whose results are cumulated in the Durability solution process.

- Add a new event to the Durability solution process.

The **Description** column for each Durability solution process displays either **NX Advanced Durability** or **NX Durability Wizard**.

- **NX Advanced Durability** indicates that the Durability commands in Advanced Simulation were used to create the process.

- **NX Durability Wizard** indicates that the Durability Wizard was used to create the process.

Name	Status	Environment	Description
Durability 1		NX Durability	NX Advanced Durability
Static Event 1	Active	NX Durability	Static Event
Load Pattern 1		NX Durability	Load Pattern
Static Event 1			
Static Event 2	Inactive	NX Durability	Static Event
Durability 1	Obsolete		
Durability 2		NX Durability	NX Durability Wizard
Static Event 1	Active	NX Durability	Static Event
Fatigue Load Variation 1		NX Durability	Load Pattern
Static Event 1			
Durability 2			

Assembly FEM support

What is it?

NX now supports Durability analyses on assembly FEMs.

Show durability or event solution log

What is it?

A new **Show Solution Log** command is now available for the Durability solution process. After you solve a durability event or a durability solution process, use this command to display the information contained in the **.ftl* file in the **Information** window.

The displayed information contains:

- The names of the following files: the **.ftl* file, the source files containing the stress or strain history, and the **.bud* files that contains the durability or event results.

- Event information: the list of strength and durability results that are calculated, evaluation and stress axis options, and excitations.

- Minimum and maximum strength and durability results.

Why should I use it?

You can view the content of the durability and event solution log stored in the **.ftl* file directly in NX. You no longer need to search the computer disk for this information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	The Durability solution process and/or the durability events must be solved.
Simulation Navigator	Right-click a Durability solution process node or an event node ® **Show Solution Log**

Create Diagnostic Groups option

What is it?

A new **Create Diagnostic Groups** option is now available in the **Durability Event Solver** dialog box and the **Durability MetaSolution Solver** dialog box. When you select this option, the durability solver creates diagnostic groups that show elements that do not have sufficient durability requirements. Two types of diagnostic groups are created:

- Event diagnostic groups

- Durability diagnostic groups

These groups are stored in the **Simulation Navigator** under the **Groups** node.

Groups
- 1 – Durability 1–Static Event 2_InvalidMaterialPropertiesGroup_Fatigue_Saftey_Factor
- 2 – Durability 1–Static Event 2_InvalidMaterialPropertiesGroup_Fatigue_Life
- 3 – Durability 1–Static Event 2_InteriorElementsGroup
- 4 – Durability 1_NoEventResultsGroup_Fatigue_Saftey_Factor
- 5 – Durability 1_NoEventResultsGroup_Fatigue_Damage
- 6 – Durability 1_NoEventResultsGroup_Fatigue_Life

The Durability solution process name appears as a prefix in the names of all the diagnostic groups. The event name is included in the names of event diagnostic groups.

The following event diagnostics groups can be created:

- *[prefix]_InvalidElementGroup* contains elements for which durability solver internal functions returned an error code. If you see this group, consider logging a problem report.
- *[prefix]_NoMaterialGroup* contains elements that do not point to any material.
- *[prefix]_IgnoredElementGroup* contains elements of types not supported by the durability solver.
- *[prefix]_IgnoredNodeGroup* contains nodes that are not processed by the durability solver because they are attached to:
  - Beam elements.
  - Elements that point to different materials.
  - Elements in one of the previously mentioned groups.
- *[prefix]_InteriorElementGroup* contains interior solid elements. Interior elements are ignored for fatigue analysis.
- *[prefix]_InvalidMaterialPropertiesGroup_[durability result type]* contains elements that do not have the necessary material properties defined for the selected durability result type.
- *[prefix]_NoResultsGroup_[durability result type]* contains elements that do not have the necessary stress or strain histories for the selected durability result type.

The following durability diagnostics group can be created:

- *[prefix]_NoEventResultsGroup_[durability result type]* contains elements that do not have the selected durability results in all participating durability events.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Solve Durability Simulation** 
Menu	**Insert** ® **Durability** ® **Solve**
Simulation Navigator	Right-click the Durability solution process node ® **Solve** Right-click the static or transient event node ® **Solve**
Location in dialog box	**Create Diagnostic Groups** check box

Support for *.rs2 files with no geometry information

What is it?

A transient event can now reference a Response Simulation results file that does not contain geometry information.

In the previous release, a transient event could only reference Response Simulation results files that contained geometry information.

Note When a transient event references a Response Simulation results file that does not contain geometry information, the durability analysis is performed over all iterations of the stress or strain histories. The **Output Start Time** and **Output End Time** boxes on the **Data Control** tab of the **Transient Durability Event** dialog box are unavailable.

Why should I use it?

You can now use previously created Response Simulation results that do not contain geometry information for a transient durability event.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability** ® **Transient Durability Event**
Menu	**Insert** ® **Durability** ® **Event** ® **Transient**
Simulation Navigator	Right-click the Durability solution process node ® **New Event** ® **Transient**
Location in dialog box	**Transient Solution List**

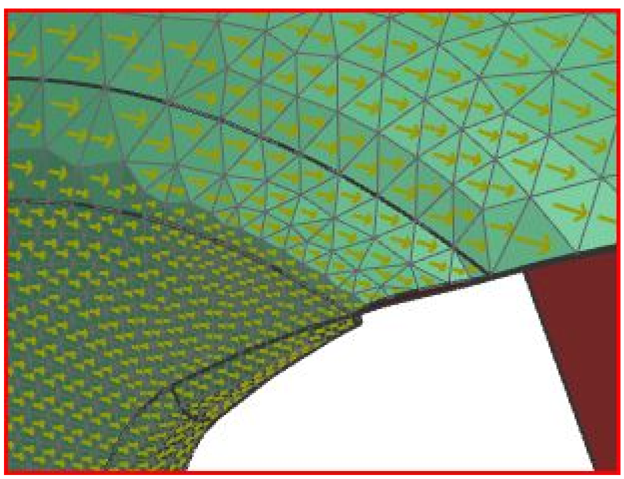
NX Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal

Preview the material orientation vector

What is it?

You can now display the material orientation vectors on shell, solid, or axisymmetric elements in the mesh, while you assign an orthotropic material in the 2D or 3D **Mesh Collector** dialog box.

Previously, you could only display the material orientation vectors using the **Model Check** command after you defined the material orientation vectors.



Preview of the material orientation of 2D shell elements

Why should I use it?

Use this option to verify the material orientation that you set.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Existing 2D or 3D meshes
Toolbar	**Advanced Simulation** ® **Mesh Collectors**
Menu	**Insert** ® **Mesh Collectors**
Simulation Navigator	Right-click a 2D or 3D mesh collector node ® **Edit**
Location in dialog box	**Material** group ® **Preview**

Laminate Composites

View the laminate physical property for a selected element

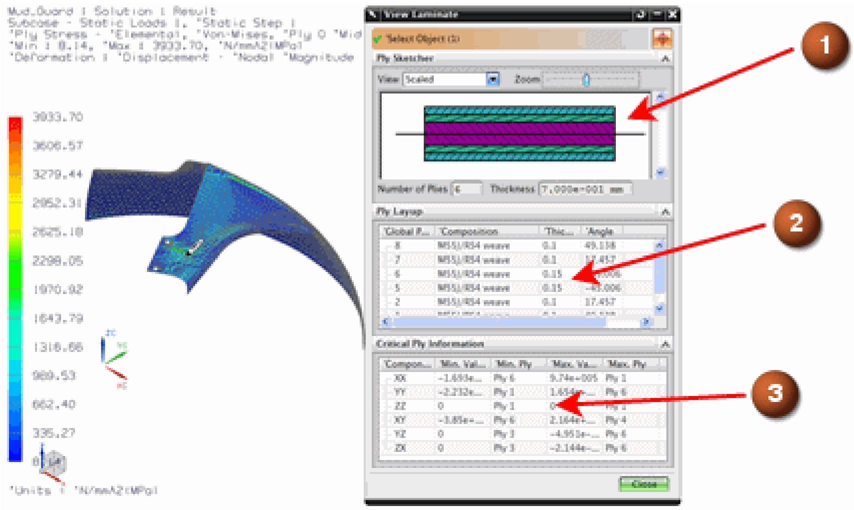
What is it?

Use the **View Laminate** command to display the laminate physical property for a selected face or element.

In the FEM file, the information is displayed for a selected polygon face or element face. All the elements in the polygon face must be in the same zone. In the Simulation file, the information is displayed for a selected element. Ply results must be displayed in the Simulation file.

The information is displayed in the command dialog box and includes the following:

- A sketch of the laminate with the number of plies and total thickness (1).
- Ply layup information that includes the global ply ID, the material composition, thickness, and angle of the ply (2).
- The minimum and maximum values of the selected results and the ply ID where the extrema occur (3). (Simulation file only)



You can double-click any of the following result types in the **Post Processing Navigator** to display the ply results in the Simulation file:

- Ply stress results
- Ply strain results
- Ply failure index results
- Bond failure index results
- Ply margin of safety results
- Ply strength ratio results

Why should I use it?

This command lets you easily view the laminate definition and critical ply results information for a selected element.

Where do I find it?

View Laminate in the FEM file

Application	Advanced Simulation
Prerequisite	Zones must be computed in the FEM file.
Simulation Navigator	Right-click the **Zones** node ® **View Laminate**

View Laminate in the Simulation file

Application	Advanced Simulation
Prerequisite	Zones must be computed in the FEM file and results displayed in Post Processing.
Toolbar	**Laminates** ® **View Laminate**

Element selection for layup offset and material orientation

What is it?

When you define a layup offset or a user-defined material orientation, you can now also select elements. Previously, you could select only polygon faces.

You can also mix polygon faces and elements in your selection.

Where do I find it?

Layup offset

Application	Advanced Simulation
Simulation Navigator	Right-click the **Layup Offset** node ® **Create User Defined Layup Offset Rule** Right-click the **Top**, **Middle**, **Bottom**, or **User defined #** node under the **Layup Offset** node ® **Edit**

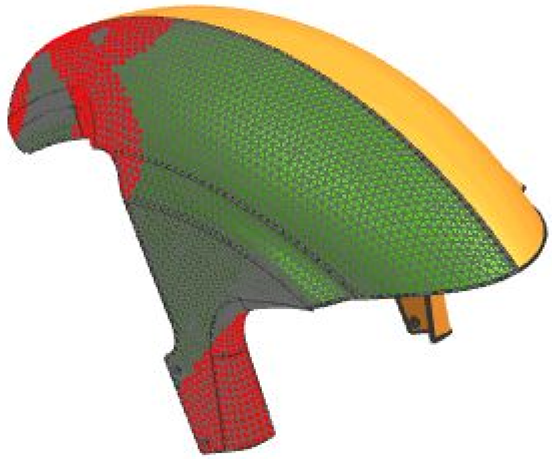
Layup material orientation

Application	Advanced Simulation
Simulation Navigator	Right-click the **Material Orientation** node ® **New Orientation** Right-click the **Orientation #** node under the **Material Orientation** node ® **Edit**

View fibers with high shear

What is it?

You can now use the **View Red Fibers** command to display the draped fibers that shear beyond the lock angle for all plies of the selected global layup.



Why should I use it?

Use this command to quickly see a problematic region over all the plies of the selected global layup. To improve the shearing of a ply, you can:

- Use a different start point and draping direction.
- Add a splice in one or more plies.
- Introduce cut curves.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click a global layup node ® **View Red Fibers**

NX FE Model Correlation

Manual mode pairing

What is it?

In addition to automatically pairing the reference and work modes, you can now manually pair modes by entering reference mode ID and work mode ID.

If you set the **Method** option to **None** in the **Automatic Pairing** group, only manual pairs are used to build the mode pairs table. You can combine manual mode pairing with automatic mode paring by selecting any pairing method other than **None**. The manual pairs you define have precedence over automatic pairs.

Why should I use it?

You have more flexibility when pairing work modes and reference modes.

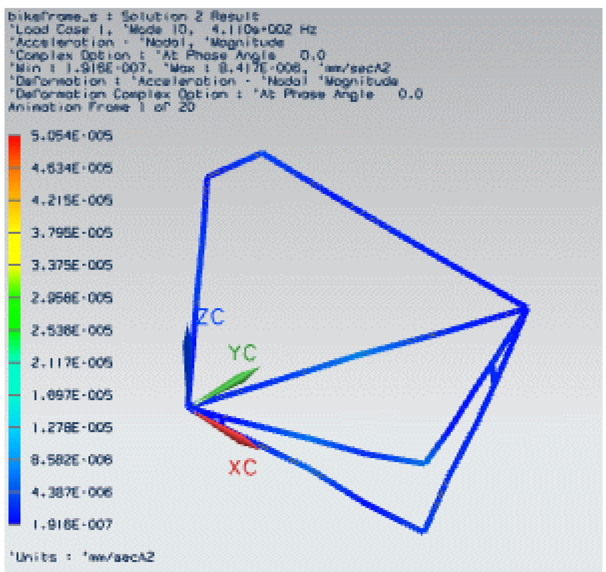
Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the **Mode Pairing** node under the **Correlation** node ® **Edit**
Location in dialog box	**Manual Pairing** group

View imported complex test mode shapes

What is it?

You can now display and animate complex mode shape results from an imported test data solution.



Complex mode 10 animated over the phase angle

Why should I use it?

Before complex mode shapes can be used in a correlation solution process, they must be converted to real mode shapes. Viewing complex test mode shapes can reveal important information. For example, while viewing the animation, you can visually assess that all nodes of the mode shape pass through the equilibrium position at about the same time. If they do, it indicates that the mode does not exhibit a high degree of complexity.

Where do I find it?

To import a test solution into new FEM and Simulation files

Application	Advanced Simulation
Menu	**File** ® **Import** ® **Simulation**
Location in dialog box	**Import** ® **MODAL TEST DATA** **Complex Mode Conversion** list ® **None**

To import a test solution into an existing analysis simulation

Application	Advanced Simulation
Toolbar	**Correlation** ® **New Test Reference Solution**
Menu	**Insert** ® **Correlation** ® **New Test Reference Solution** or **Insert** ® **Solution**
Simulation Navigator	Right-click the Simulation node ® **New Solution**
Location in dialog box	**Solver** ® **MODAL TEST DATA** **Complex Mode Conversion** list ® **None**

Complex mode conversion options

What is it?

Multiple methods for converting complex mode shapes to real mode shapes are now available from the **Complex Mode Conversion** list. You can select a method when you import, create, or edit the test data solution.

None — When you select this method, NX imports complex modes without converting them to real modes. Use this method to display complex modes in Post Processing.

Signed Amplitude — When you select this method, the correlation solver uses the signed amplitude method to convert the complex modes into real modes.

Real — When you select this method, the correlation solver uses the real part of the complex modes for the calculations.

Imaginary — When you select this method, the correlation solver uses the imaginary part of the complex modes for the calculations.

Previously, NX always converted the complex modes into real modes using the signed amplitude method.

Where do I find it?

To import a test solution into new FEM and Simulation files

Application	Advanced Simulation
Menu	**File** ® **Import** ® **Simulation**
Location in dialog box	**Import** ® **MODAL TEST DATA** **Complex Mode Conversion** list

To import a test solution into an existing analysis simulation

Application	Advanced Simulation
Toolbar	**Correlation** ® **New Test Reference Solution** 
Menu	**Insert** ® **Correlation** ® **New Test Reference Solution** or **Insert** ® **Solution**
Simulation Navigator	Right-click the Simulation node ® **New Solution**
Location in dialog box	**Solver** ® **MODAL TEST DATA** **Complex Mode Conversion** list

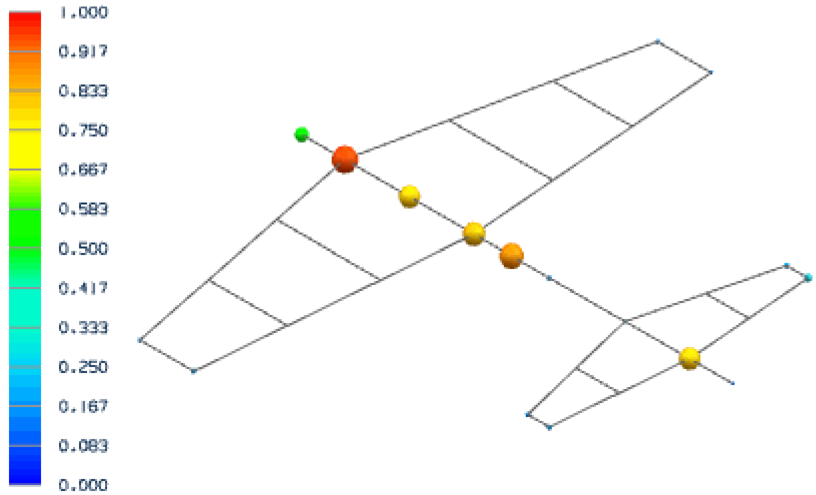
To edit an existing test solution

Application	Advanced Simulation
Simulation Navigator	Right-click the test solution node ® **Edit**
Location in dialog box	**Complex Mode Conversion** list

Display COMAC results in Post Processing

What is it?

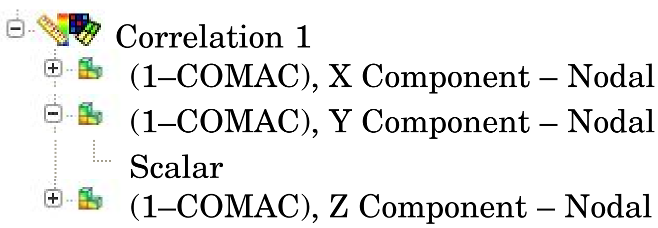
You can now display (1 – COMAC) results in the graphics window.



(1–COMAC) results for the Y component

The results displayed are (1–COMAC) values at each translational degree-of-freedom, not COMAC values. Thus, values approaching 1.0 indicate that these degrees-of-freedom have poor correlation across the set of paired mode shapes.

After you generate (1 – COMAC) results, you get the following results nodes in the **Post Processing Navigator**:



Why should I use it?

You can identify graphically the parts of the structure which are responsible for the low degrees of correlation between reference solution and work solution.

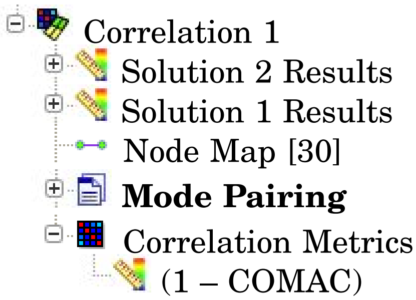
Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the **Correlation Metrics** node under the **Correlation** node ® **Generate (1 – COMAC) Results**

Correlation Metrics node in Simulation Navigator

What is it?

A new **Correlation Metrics** node is displayed under the **Correlation** node in the **Simulation Navigator**.



The following commands are available when you right-click the **Correlation Metrics** node.

Icon	Command name	Description	Comments
	Correlate	Opens the **Correlate** dialog box to let you select, evaluate, and export the correlation metric.	In previous releases, this command was a right-click command for the **Correlation** node.
	Generate (1 – COMAC) Results	Generates the **(1 – COMAC)** results node under the **Correlation Metrics** node. The **(1 – COMAC)** results node contains the (1 – COMAC) results that you can display in the graphics window in Post Processing.	For more information, see **Title not found**.

UNV dataset 2414 support

What is it?

NX FE Model Correlation now supports UNV files that contain dataset 2414 for mode shape result data. Dataset 2414 is often written together with the geometry datasets in a single UNV file.

When you import or create modal test data solutions, and the UNV file contains both the geometry datasets and dataset 2414, in the **Import Modal Test Data** dialog box or the **Solution** dialog box, specify the UNV file in the **Geometry File** box and leave the **Modes File** blank.

Why should I use it?

The NX FE Model Correlation support of the dataset 2414 improves compatibility with Test for I-deas and other test software.

Where do I find it?

To import a test solution into new FEM and Simulation files

Application	Advanced Simulation
Menu	**File** ® **Import** ® **Simulation**
Location in dialog box	**Import** ® **MODAL TEST DATA**

To import a test solution into an existing analysis simulation

Application	Advanced Simulation
Toolbar	**Correlation** ® **New Test Reference Solution**
Menu	**Insert** ® **Correlation** ® **New Test Reference Solution** or **Insert** ® **Solution**
Simulation Navigator	Right-click the Simulation node ® **New Solution**
Location in dialog box	**Solver** ® **MODAL TEST DATA**

To edit an existing test solution

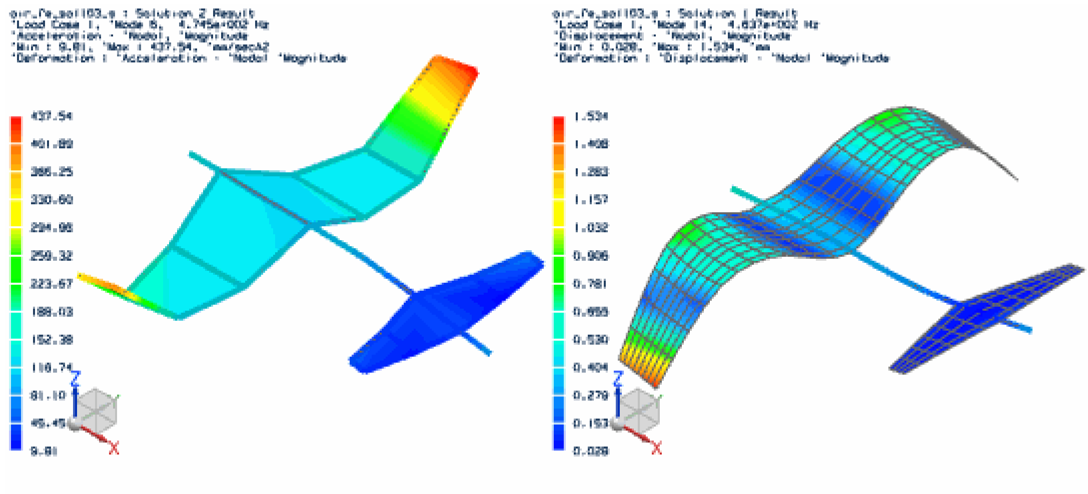
Application	Advanced Simulation
Simulation Navigator	Right-click the test solution node ® **Edit**

Phase adjustment for side by side mode shape display

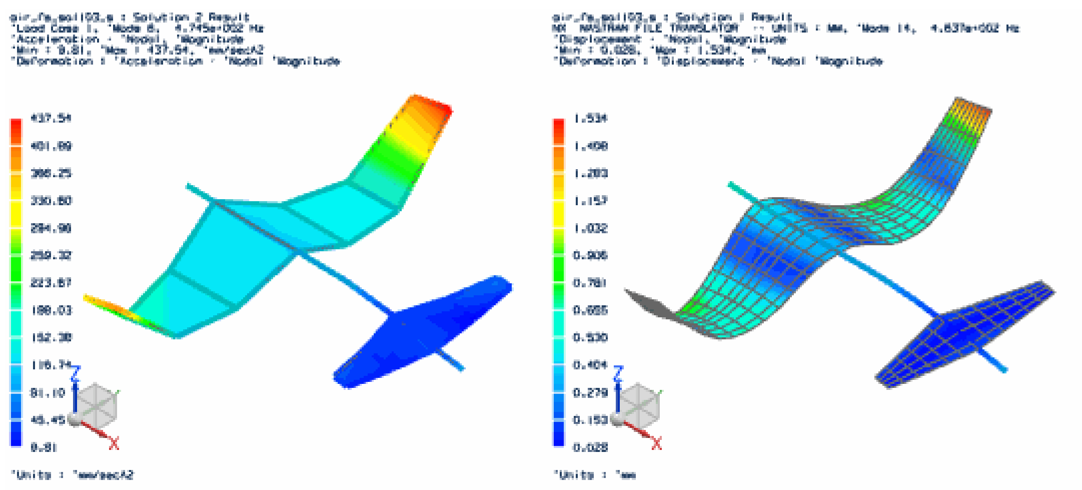
What is it?

NX now automatically adjusts the phases when you use **Side-by-Side Display** and **Side-by-Side Animation** commands on mode pairs in the **Correlation Details View** subpanel.

To achieve this, NX negatively scales the work mode shape deformation if the modal scale factor between the modes is negative.



Out-of-phase display of a mode pair



Automatically adjusted phase display of a mode pair

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	**Mode Pairs [#]** node ® **Correlation Details View** subpanel ® right-click one or more mode pairs ® **Side-by-Side Display/Side-by-Side Animation**

NX FE Model Updating

MSC Nastran DESOPT 200 solution support

What is it?

You can now select the MSC Nastran DESOPT 200 — Model Update solution as your work solution for the Model Update solution process. In previous releases, you could only select the NX Nastran DESOPT 200 — Model Update solution.

MSC Nastran version 2007.0 or 2008.0 are supported.

Note A MSC Nastran DMAP license is required.

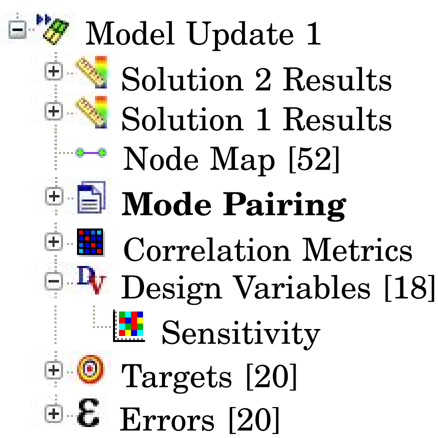
Where do I find it?

Application	Advanced Simulation
Toolbar	**Model Update** ® **New Model Update**
Menu	**Insert** ® **Model Update**
Simulation Navigator	Right-click the simulation node ® **New Solution Process** ® **Model Update**
Location in dialog box	**Work Solution (Analysis)** group ® **Type** column ® MSC **DESOPT 200 – Model Update** solution

Sensitivity node in Simulation Navigator

What is it?

A new **Sensitivity** node is displayed under the **Design Variables [#]** node in the **Simulation Navigator**.



The following commands are available when you right-click the **Sensitivity** node.

Icon	Command name	Description	Comments
	**Plot Sensitivities**	In the **Sensitivity Matrix** dialog box, plots the design variable sensitivities with respect to all optimization targets.	In previous releases, this command was a right-click command for the **Design Variables [#]** node.
	**Export to Excel Spreadsheet**	Exports sensitivity values to an Excel spreadsheet.	These are new commands in this release.
	**Export to CSV File**	Exports sensitivity values to a CSV file.	
	**Information**	Opens the **Information** window that lists the sensitivity values.	

Chapter

7 *Teamcenter Integration for NX*

Performance improvement during part loading

What is it?

Multi-threading has been introduced in NX 7.5.1 into the file download process. You can now load parts faster when the part files do not already exist in the FMS client cache.

Though part loading performance depends upon several factors, the largest improvement in performance happens when the following conditions are met:

- The deployment configuration is four-tier
- The network has high latency and low bandwidth
- There is no local Teamcenter volume

A typical scenario is loading an assembly for the first time where no corresponding NX part files exist in the FMS client cache.

Why should I use it?

Performance is improved when loading NX part files.

Where do I find it?

Performance improvement is on by default and is activated with every NX part load operation.

Application	Teamcenter Integration
Menu	**File**® **Open** and other operations where you are loading part files.

Occurrence effectivity in Teamcenter and NX

What is it?

Occurrence effectivity defined in Teamcenter is applied to assemblies loaded in NX. Occurrences that are configured out of an assembly in Teamcenter are suppressed when you load the assembly in NX.

Occurrence effectivity is typically used to track major changes in the evolution of a product. This is different from revision effectivity which is typically used to track smaller changes in a product. Occurrence effectivity is used to track the dates of effectivity for a part.

Occurrence effectivity is set in Teamcenter and cannot be changed in NX; parts that are suppressed in NX when an assembly is loaded cannot be unsupressed in NX. You can perform normal modeling operations that include the suppressed components and when you save an assembly that contains suppressed components, they are saved as part of the assembly in Teamcenter.

To see the components that are suppressed, in the **Assembly Navigator**, you can right-click on a column heading and select **Include Suppressed Components**.

The revision rule used (defined in Teamcenter) and the date in its definition determines the occurrence effectivity applied. You can further control the occurrence effectivity by changing the date value in NX for the revision rule. If you cannot change the date, then **Today** is implied. Changing the revision rule date in NX creates a transient revision rule.

When you use the **Export Assembly outside Teamcenter** command, occurrence effectivity is ignored; suppressed components are considered a direct part of the assembly and included in the operation.

For additional information on setting occurrence effectivity, see the Teamcenter documentation.

Why should I use it?

You can control occurrence effectivity in Teamcenter and apply it to assemblies loaded in NX.

Save As a non-master drawing to any Item Revision

What is it?

When you use **Save As** on a non-master drawing, you can save it to any existing Item Revision of the same Item. This allows you to create an initial drawing with a UGMaster dataset in an Item Revision, make several revisions to the model, then save the drawing to a later Item Revision.

In the **Save Part File As** dialog box, the **Revision** box is no longer automatically set to the existing revision and grayed out. The **Revision** box is now modifiable and you can enter any existing Item Revision.

A message prompting you to manually update the drawing to the UGMaster of the new Item Revision is displayed. The update is also done automatically when you open up the drawing in the new Item Revision if you have the **Update Master Model** customer default selected.

You cannot save a non-master drawing to a Item Revision of a different Item. However, you can use Save As to save a non-master drawing as a UGMaster of a completely new Item/Item Revision and keep the master-model relationship to the model in the source Item/Item Revision.

Note You cannot save a drawing as a UGMaster and keep the master-model relationship if you save to a new Item/Item Revision and a new Item Type.

Why should I use it?

You can save a non-master drawing to a different Item Revision of the same Item.

Where do I find it?

Application	Teamcenter Integration
Menu	**File® Save As**
Location in dialog box	**Revision**

Browse to a Teamcenter location when cloning or importing

What is it?

When you specify the Teamcenter location when you clone an assembly or import an assembly, you can now browse to the location in Teamcenter instead of manually typing the path.

In the **Clone Assembly** and **Import Assembly** dialog boxes, you can click the **Browse** button for the **Default Output Folder** and select the folder in the **Folder Select** dialog box.

The **Browse** button allows you to view your **Home** node and folders under it. You can specify a folder outside of your **Home** by manually entering it in the text box, but you must have write access to the folder.

Why should I use it?

Browsing for the folder results in fewer errors than when you manually type the path.

Where do I find it?

Application	Teamcenter Integration
Menu	**Assemblies® Cloning® Create Clone Assembly** **File® Import Assembly into Teamcenter**
Location in dialog box	**Clone Assembly** dialog box® **Numbering** tab® **Default Output Folder** box® **Browse** button **Import Assembly** dialog box® **Numbering** tab® **Default Output Folder** box® **Browse** button

Show revisions of a replacement component in an assembly

What is it?

When you are replacing a component in an assembly, you can show the additional revisions of the component in the dialog box without browsing or doing a part file search to find the revisions.

In the **Replace Component** dialog box, you can select a component, and then click **Show Revisions** to display the additional revisions of the part in the **Unloaded Parts** list. You can then select a revision and replace the component with that revision.

Note You can select multiple components and click **Show Revisions**. All of the available revisions for all of the components are displayed.

A **Browse** button is available under the **Unloaded Parts** list so you can browse or search for another revision that you want to use as a replacement. If all of the revisions are already listed in **Loaded Parts** or **Unloaded Parts**, the **Browse** button is disabled.

Why should I use it?

You can easily find additional revisions for a component in an assembly and use one of them to replace an existing revision.

Where do I find it?

Application	Teamcenter Integration
Menu	**Assemblies® Components® Replace Component**
Assembly Navigator	Right-click a component ® **Replace Component**
Location in dialog box	**Replacement Part** group

Drawing booklets displayed in Teamcenter Navigator

What is it?

Drawing booklets are shown in **Teamcenter Navigator** as objects. A drawing booklet contains related drawings and drawing sheets that are grouped together. The booklets are typically used for very large assemblies or complex parts that require a large number of drawings.

A drawing booklet contains Items and Item Revisions. Under the Item Revision are containers that represent the drawings and drawing sheets. A booklet can be located under **Home** or in any folder where you have write access.

The **Create Automated Drawings** wizard determines the grouping of drawings, location of the drawing booklets in **Teamcenter Navigator**, and how containers are identified and arranged within the drawing booklets.

You can use the NX **Part Navigator** to get a more complete view of the drawing booklet.

For additional information on the creation and use of drawing booklets, see Drawing Booklets.

Why should I use it?

Drawing booklets make it easier to manage a large quantity of drawings and drawing sheets.

Where do I find it?

Applcation	Teamcenter Integration
Teamcenter Navigator	Drawing booklets are objects in the **Home** folder or sub-folders.

Chapter

8 *Inspection and Validation*

Check-Mate

Updates to Check-Mate checkers

What is it?

Sixteen new checkers are added in the following three categories:

Modeling
Get Information
SASIG-PDQ test

Checker category	New Checker	Description
Modeling – Features	Check If The Part File is Saved with Rollback Data	Checks whether a part file is saved with rollback data.
Get Information – Modeling	Report Features Without Dependencies	Reports features without any dependencies in a file. An independent feature does not have any parent or child feature.
Get Information – Modeling	Report Features Alert Messages	Reports alert messages from the features.
Get Information – Drafting	Report Information About Section View	Reports available information about the section view.

SASIG-PDQ test categories now include all 64 geometric quality criteria contained in SASIG PDQ guidelines version 3.

Category	New Checkers
Curves (G-CU)	• Fragmented Curve: G-CU-FG • Curve with a small radius of curvature: G-CU-CR • Inappropriate degree linear curve: G-CU-ID
Surface (G-SU)	• Multi-face Surface: G-SU-MU • Inappropriate degree planar surface: G-SU-ID

Category	New Checkers
Faces (G-FA)	• Analytical Face: G-FA-AN • Closed face: G-FA-CL • Inconsistent face on surface: G-FA-IT
Edges (G-ED)	• Analytical edge: G-ED-AN • Inconsistent edge on curve: G-ED-IT
Shells (G-SH)	• Over-Used vertex: G-SH-OU
Solids (G-SO)	• Intersecting Shells: G-SO-IS

The following checkers are changed:

DFM checkers

Because the DFM checkers are not intended to be generic checkers for any other sheet metal operations, to prevent inappropriate use the DFM checkers are moved to a new DFM example files folder, and the DFM checkers do not appear in the out-of-the-box checkers list. The new default folder location for the DFM checkers files is *%UGII_BASE_DIR%\DESIGN_TOOLS\checkmate\examples\dfm*

To use these DFM checkers, copy them to a directory and assign that full directory path (without a trailing slash) to the Check-Mate environment variable **UGCHECKMATE_USER_DIR**. When NX is subsequently started, the DFM checkers should be visible in a **DFM** category inside the Check-Mate list of available checks. Please refer to *Check-Mate system administration* in the *NX Help* document for details of the configuration process.

Why should I use it?

The new and updated checkers help you to better validate the quality of product data.

Where do I find it?

Resource bar	**HD3D Tools**® **Check-Mate**
Toolbar	**Check-Mate**
Menu	**Analysis**® **Check-Mate**® **Set Up Tests** **Analysis**® **Check-Mate**® **Author Tests**
Location in dialog box	(**Set Up Tests**) **Tests** tab® **Categories** group (**Author Tests**) **Create/Edit Profile** tab® **Categories** group
File system	*%UGII_BASE_DIR%\DESIGN_TOOLS\checkmate\examples\dfm*

Updates in Check-Mate KF functions

What is it?

New SASIG-PDQ checking functions and new modeling checking and reporting functions are added for the new Check-Mate checkers.

You can use the part attribute functions in checkers to get the following part data:

- Attribute values from attribute types of integer, number, string, boolean, null, and date.
- Category
- Dimension and Unit of number type attribute values

The new functions include the following:

mqc_askPartAttributeValues
mgc_askPartAttributesOfTypes
mqc_askPartAttributesOfUnits
mqc_askPartAttributesOfCategories

Why should I use it?

The new functions enable you to gather more kinds of data from a part, and thus create new kinds of checks.

Where do I find it?

You can use the new functions when creating or editing checkers. You can use the **Author Tests** dialog box for the steps required to author Check-Mate tests.

Resource bar	**HD3D Tools® Check-Mate**
Toolbar	**Check-Mate**
Menu	**Analysis® Check-Mate® Author Tests**
Location in dialog box	**Create/Edit Checker** tab

Updates in Check-Mate examples

What is it?

NXOpen Check-Mate checker source code file examples are added in the examples folder.

Why should I use it?

The new examples help you to understand how to create NXOpen functions for Check-Mate checkers, and thus create new kinds of checks.

Where do I find it?

File system	*%UGII_BASE_DIR%\DESIGN_TOOLS\checkmate\examples\NXOpenExamples*

CMM Inspection Programming

General enhancements

Link to PMI enhancements

What is it?

When you link to PMI (Product and Manufacturing Information) data, an information window reports on all Geometric Dimensioning and Tolerancing (GD&T) and dimensional PMI markup linked to CMM Inspection Programming.

In the **Inspection Navigator**:

- Items marked as successfully linked appear in their appropriate feature, tolerance, and path locations.

- PMI markup not supported or used by CMM Inspection Programming, such as notes and labels, do not appear.

- Items marked as linked but incomplete also display the ⊘ icon. These may require some manual modification, such as clearly linking the markup to new or existing inspection features, to reestablish links between markup and inspection features.

Why should I use it?

The PMI report lets you quickly confirm that your PMI markup has successfully linked to CMM Inspection Programming. In **Inspection Navigator**, you can compare the results with the new inspection features and tolerances and make changes in CMM Inspection Programming, PMI, or both.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must define feature-based tolerance annotations in *PMI* and create an inspection file.
Toolbar	**Operations® Link to PMI**
Menu	**Tools® Inspection Navigator® Operation® Link to PMI**

Inspection Navigator enhancements

What is it?

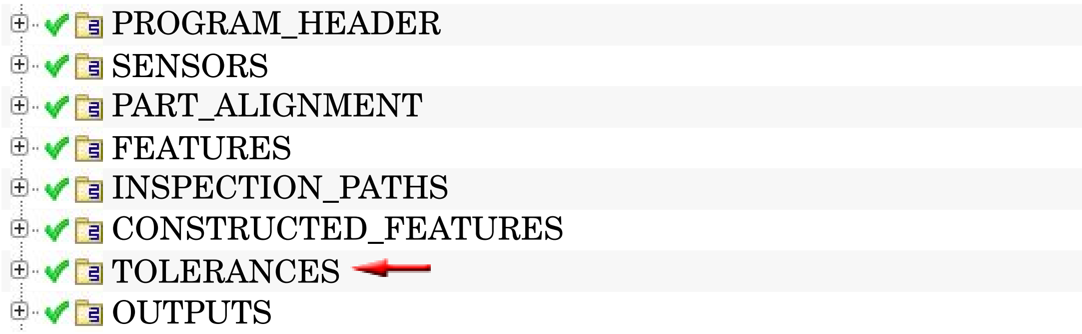
There are many enhancements to the **Program Order** view of the **Inspection Navigator**.

- You can reorder inspection paths and other elements.

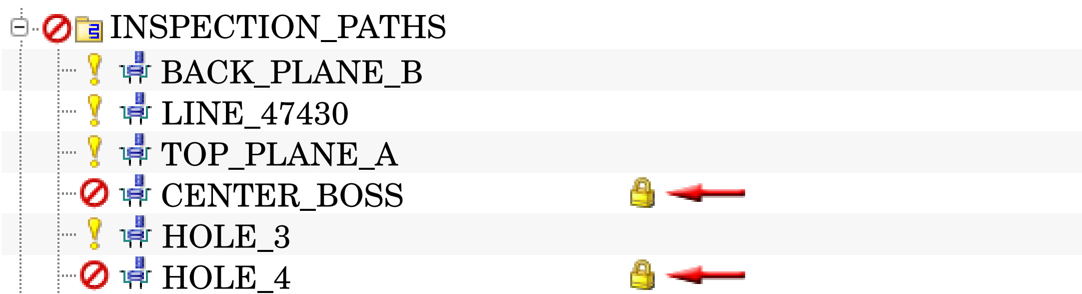
  - You can click the element to be moved and then drop it on the element that it should follow.

  - You can select multiple elements by pressing Ctrl, and then drag them to a new location. If you drag them into a group, they become the last element in that group.

- Measured and constructed features can correctly reference datum definitions. In the **DMIS General Setup** template, the **TOLERANCES** group follows the **INSPECTION_PATHS** and **CONSTRUCTED_FEATURES** groups.



- You can lock an inspection path in the **Program Order** view of the **Inspection Navigator** by right-clicking the path to be locked and choosing **Object® Lock**.



- The following elements now display the **Out of Date** icon to indicate that they are not yet generated for inclusion in the program.

  o Inspection features with modified underlying geometry.

  o Tolerances whose underlying definitions have changed.

  If any features or tolerances are out of date, then their parent groups also display **Out of Date** icons. Parent groups include **FEATURES**, **TOLERANCES**, and **INSPECTION_PATHS**, and as well as the **CMM_INSPECTION_PROGRAM** header.

Why should I use it?

- Locking an inspection path ensures that no changes are accidentally made to custom settings.

- Reordered program groups ensure that datum references to features and constructed features can exist without program errors. You no longer need to manually cut and paste datums to a higher group in the **Inspection Navigator**.

- The **Out of Date** icon for inspection features and tolerances now directly indicates that you must regenerate your inspection program.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	An inspection file with an inspection feature and an inspection path on the inspection feature.
Resource bar	**Inspection Navigator**
Toolbar	**Navigator® Program Order View**
Menu	**Tools® Inspection Navigator® View® Program Order View**

Inspection feature enhancements

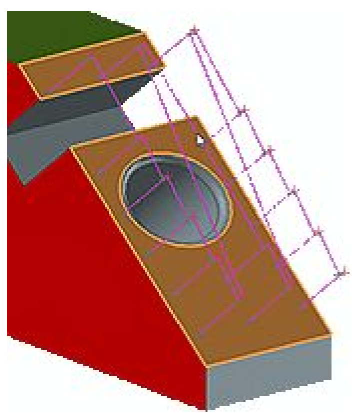
What is it?

- When creating a plane inspection feature, you can now select multiple faces as long as they are coplanar.

- The **Plane Inspection Feature** and **Cone Inspection Feature** dialog boxes now include an **Extent** subgroup that lets you specify whether a plane or cone inspection feature created with face geometry is **Bounded** or **Unbounded**.

Plane features with multiple coplanar surfaces

When you create inspection paths on a plane feature that consists of multiple coplanar surfaces, the path automatically avoids any voids or obstructions in the surfaces.

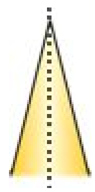


Note You cannot automatically generate a plane feature on a part with coplanar surfaces using the **Link to PMI** command.

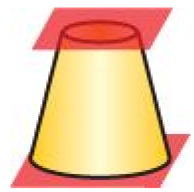
Bounded versus unbounded features

While bounded and unbounded features do not appear different from other plane and cone features in the NX graphics window, bounded features differ from unbounded features in that they have finite planar extent boundaries. These generate `BOUND` events in your program output.

Any older inspection files containing planes or cones that were defined using face geometry will be automatically set to bounded when you generate and post process them.



Unbounded cone



Bounded cone

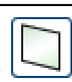
```
BOUND/F(TruncatedCone),F(Top),F(Bottom)
```

Why should I use it?

- The ability to create a single feature from coplanar geometry lets you quickly create a single plane feature instead of many plane features, and to apply a single tolerance for all of them. Less features and tolerances in the **Inspection Navigator** make your workspace easier to organize, modify, and inspect.

- While plane and cone inspection features are unbounded by definition, tolerances such as position and total runout require bounded features.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection file.
Toolbar	**Feature® Plane**  **Feature® Cone** 
Menu	**Insert® Feature® Plane** or **Cone**
Inspection Navigator	Right-click any node and choose **Insert® Feature® Plane** or **Cone**

Enhancements to constructed features

What is it?

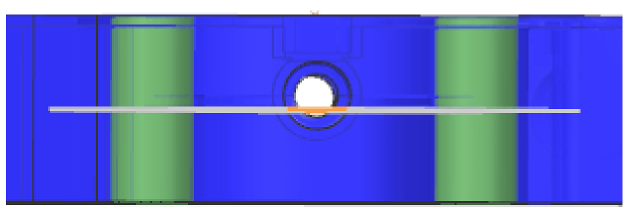
Constructed features are mathematically constructed from the characteristics of standard inspection features. The **Constructed Features** dialog box has been reorganized and enhanced to let you:

- Select a nominal feature, now termed **Design Feature**, prior to selecting the nominal and actual inspection features that define it.
- Create a new design feature for most types of constructed plane features without selecting a nominal inspection feature.
- Filter standard inspection features to be selected by feature type, and specify whether they should use actual or nominal values.

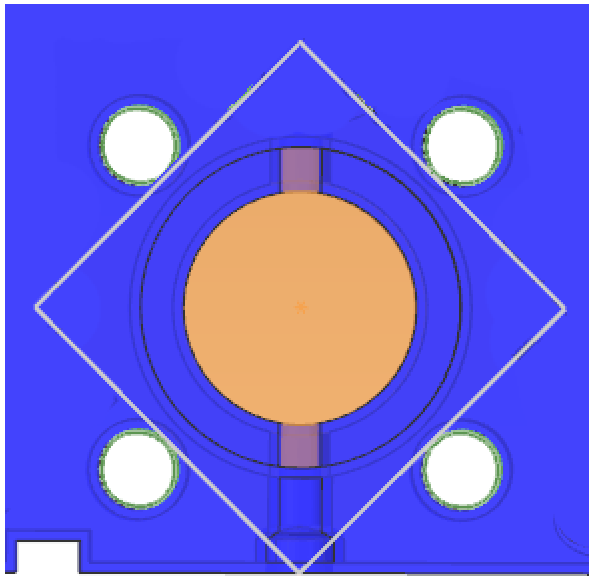
You can also select from a wide range of new and modified methods, now termed *construction forms*.

Construction form, design feature type	Description
Best Fit Point, line, plane, arc, circle, cylinder, cone, or sphere	Determines the best fit through or within any number of previously defined, point-reducible inspection features. If multiple solutions exist, the feature is constructed as close to the design feature as possible.

Example The following shows a constructed plane feature in which the design feature does not use an existing inspection feature for its nominal. Using the actuals of four cylinder features, the calculated plane rests on the axial center points of the cylinders. Although it appears to be bounded by the cylinders in the graphics window, the constructed feature is actually unbounded.

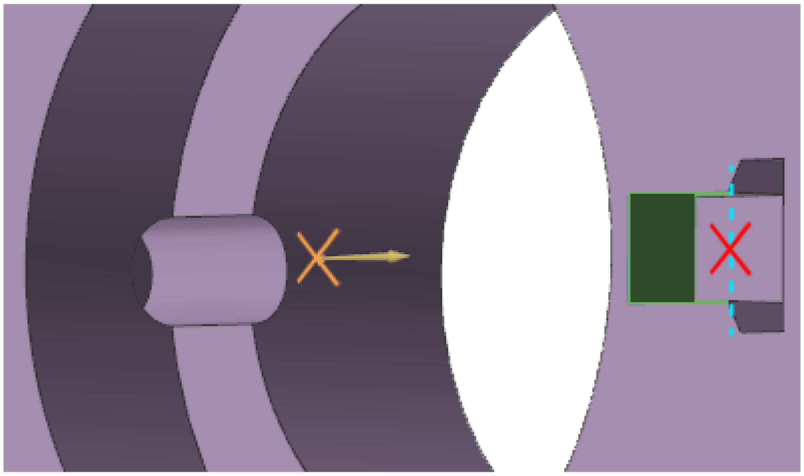


Front



Top

Transform Point, line, plane, arc, circle, cylinder, cone, or sphere	Transforms the constructed feature using the geometric attributes of a measured base feature and the following: • The current Part Coordinate System (PCS) • A nominal or actual PCS created using the **Alignment** command
Minimum, **Maximum** Point	Using a selected **Sub-Feature1** as an actual reference, creates a constructed point feature at the minimum or maximum distance to the subfeature in one of the following ways: • Along a specified vector • Radially from the center of an arc, circle, cylinder, cone, or sphere feature • Using the ijk of another specified feature **Example** The following shows a constructed point feature that uses a nominal point at the center of a large circle as the design feature. The constructed feature uses the vector direction of the green plane feature, as well as the maximum distance of a closed slot feature. Actuals are derived from both the plane and the slot.



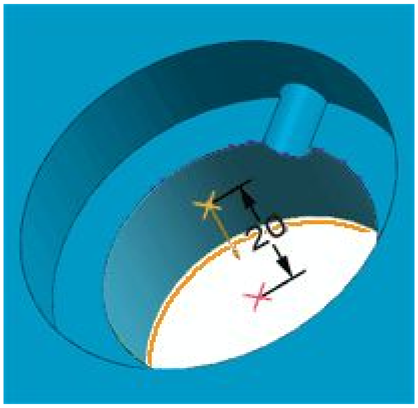
Move By Feature, **Move By Vector**

Point

Move by Feature creates a constructed point feature a specified distance from a nominal design feature, using the vector direction of another selected feature. Actuals are derived from the direction feature, a different measured point-reducible feature, or both.

Move by Feature works similarly but uses a specified, nominal direction vector rather than a direction feature.

Example The following shows a constructed point feature moved by vector. A nominal point design feature, in orange, is projected 20 millimeters along a specified direction vector, using a point-reducible measured circle feature as the actual.

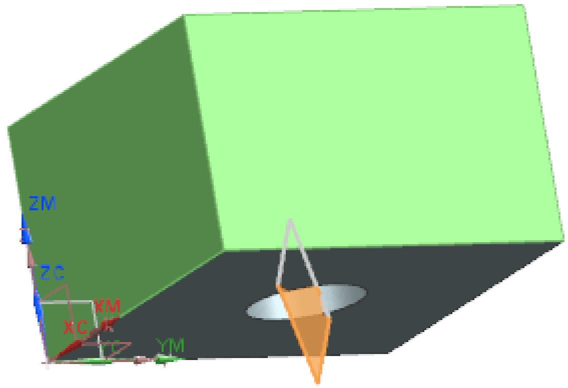


Middle

Point, line, or plane

Creates a constructed point, line, or plane feature at the midline, midplane, or midpoint between two inspection features.

Example Below is a constructed plane feature in which the design feature does not use an existing inspection feature for its nominal. The plane design feature, in orange, lies midway between two plane features, both of which are actuals. Although it appears to be bounded, the constructed feature is actually infinite.

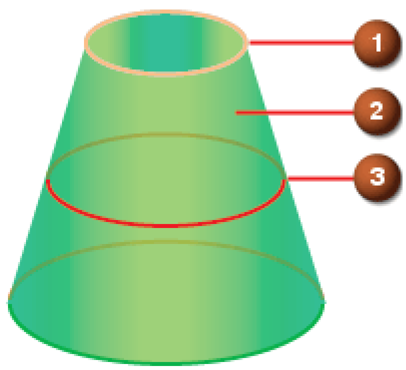


Cone Diameter

Circle

Creates a constructed circle feature at the location of the actual cone where the cone has the specified **Diameter** value.

Example



1	The design circle feature, with a nominal diameter of 8.0 millimeters.
2	The actual cone feature. A circle feature at the base has a diameter of 15.5.
3	The constructed circle feature. Since a

Diameter of 11.75 is specified, the constructed circle is midway along the cone:

```
(15.5 - 8) / 2 = 3.75
```

and

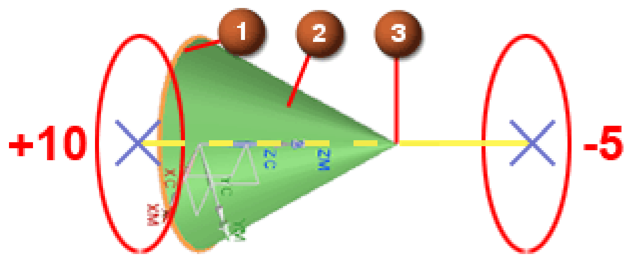
```
3.75 + 8 = 11.75)
```

Cone Distance

Circle

Creates a constructed circle feature at a specified distance from the actual cone vertex along the actual cone axis toward the base of the cone. A negative distance creates the constructed circle in the opposite direction of the actual cone axis.

Example



1 Design feature

2 Actual cone feature

3 Vertex and point of distance origin on actual cone axis

✕ Positive and negative constructed circle feature locations, based on **Distance**

Why should I use it?

Constructed features are often used to measure the size, location, and orientation of a theoretical feature or group of features relative to another feature or group of features. Use constructed features to derive a single feature from the point, line, and plane characteristics of other features.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection file as well as any inspection features you need to create the constructed feature.
Toolbar	**Insert® Constructed Feature** 
Menu	**Insert® Constructed Feature**
Inspection Navigator	Right-click any node and choose **Insert® Constructed Feature**

New DMIS 3.0 postprocessor available

What is it?

When you postprocess an inspection program, CMM Inspection Programming now offers the ability to use the 3.0 Dimensional Measuring Interface Specification (DMIS) standard, in addition to the recently approved DMIS 5.2 version. The 3.0 version is the second DMIS standard, after 2.1, to be submitted to ANSI and accepted as an American National Standard.

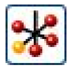
Why should I use it?

Older machines and systems that do not support the 5.2 standard can use the 3.0 standard.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection program.
Toolbar	**Operations® Post Process** 
Menu	**Tools® Inspection Navigator® Output® Inspection Post Process**
Inspection Navigator	Right-click a node to be postprocessed and choose **Post Process**

Machine simulation enhancements



Sensor command

What is it?

Use the **Sensor** command to create personalized sensor definitions for each probe in your workflow.

Touch probe sensors are used with dimensional measurement equipment to gather measurement data. CMM Inspection Programming automatically creates a best sensor for gathering measurement data based on the machine head and probe combination used in an inspection path. However, you can also create a custom sensor definition for which you define the probe tip, angle combination, and unique name.

This example shows a sensor defined for an inspection path on the left side of the part. To avoid potential collision with the part, the head (B) rotates 90° around the Z-axis, and the barrel (A) rotates 45° around the X-axis.

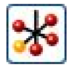
Why should I use it?

Sensor definitions let you:

- Uniquely name each sensor as your workflow or standards require.
- Define a different sensor for each stem on a multi-tip probe, or different sensors for any probe.

- Specify default A and B angles to indicate how the probe should be positioned or rotated.

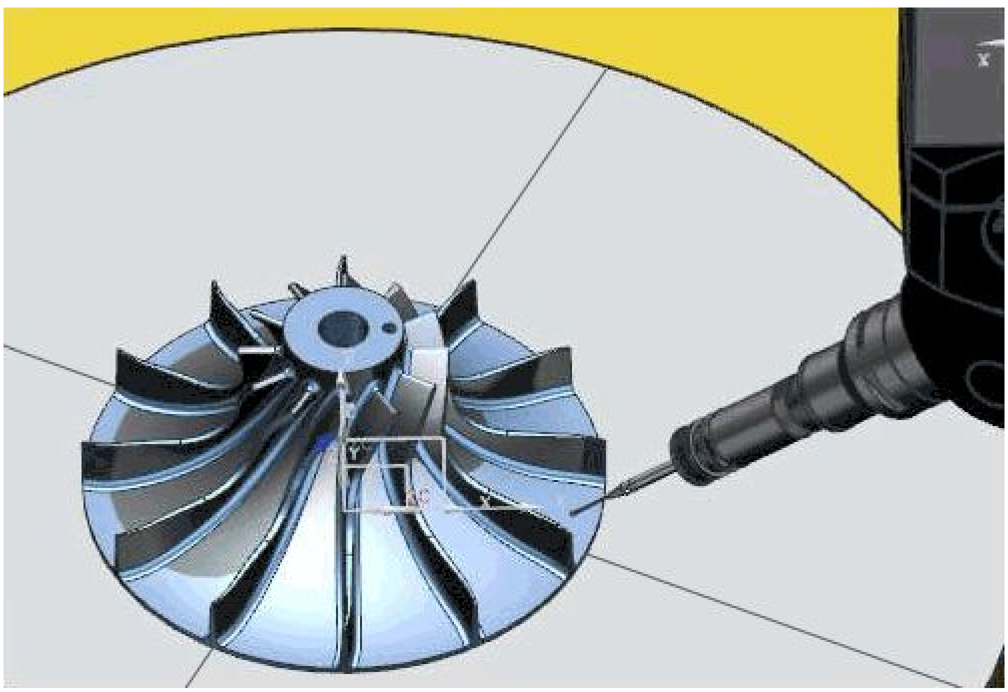
Where do I find it?

Application	CMM Inspection Programming
Prerequisite	An inspection file with at least one probe loaded, as described in the NX 7.5 CMM Inspection Programming help.
Toolbar	**Insert® Sensor** 
Menu	**Insert® Sensor**
Inspection Navigator	Right-click any node and choose **Insert® Sensor**

Rotate table command

What is it?

Use the **Rotary Table** command to rotate a work part on a rotary table system integrated with your CMM.



You can also define a second table either dependent on or independent of the first table to further increase the flexibility of your inspection process and reduce the machine time needed to calibrate probe angles.

Why should I use it?

A rotary table can:

- Measure circular patterns of features on a complex work part while maintaining coordinate system integrity.

- Provide probe access to blocked areas that might otherwise require multiple setups to inspect.

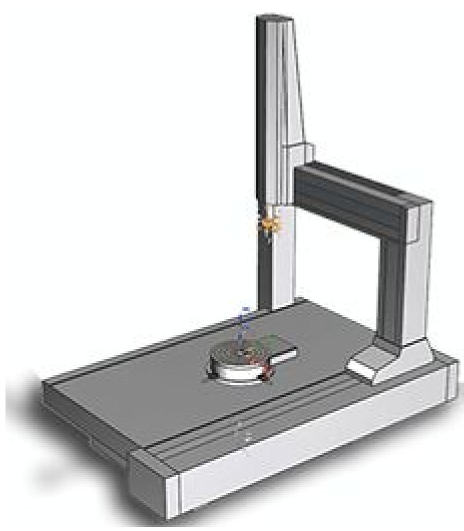
Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection file and load a virtual machine that contains a rotary table.
Toolbar	**Insert® Rotate Table**
Menu	**Insert® Rotate Table**
Inspection Navigator	Right-click any node and choose **Insert® Rotate Table**

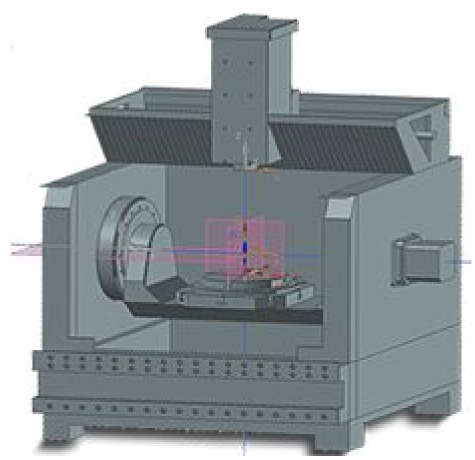
New virtual probe and machine models

What is it?

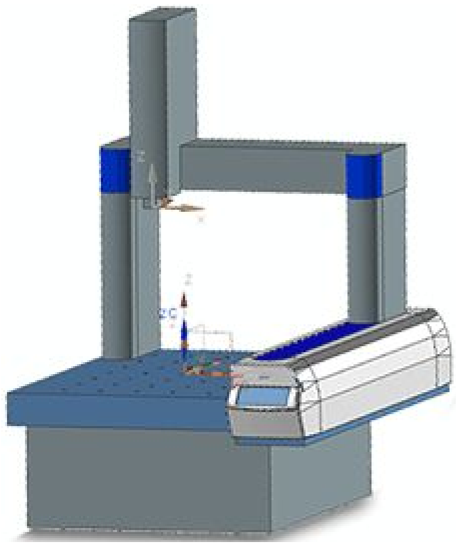
The CMM Inspection Programming library features several new machine models, as well as a new probe and head.



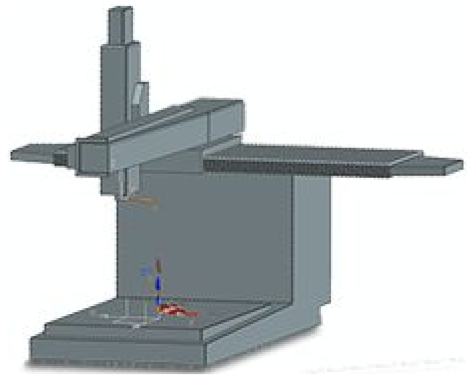
Brown and Sharpe X-cel 122010 with rotary table



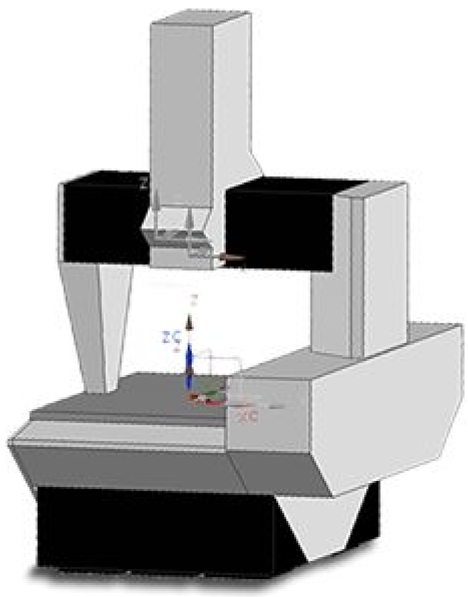
Zeiss Duramax



Hermle dual-axis rotary machine



Zeiss Prismo



Zeiss Contura

Star probe mounted to VAST fixed head

Why should I use it?

New machine models offer more options when you run inspection simulations. For detailed information on how to load machine components, see *Load a virtual CMM machine* in the NX 8.0 online help.

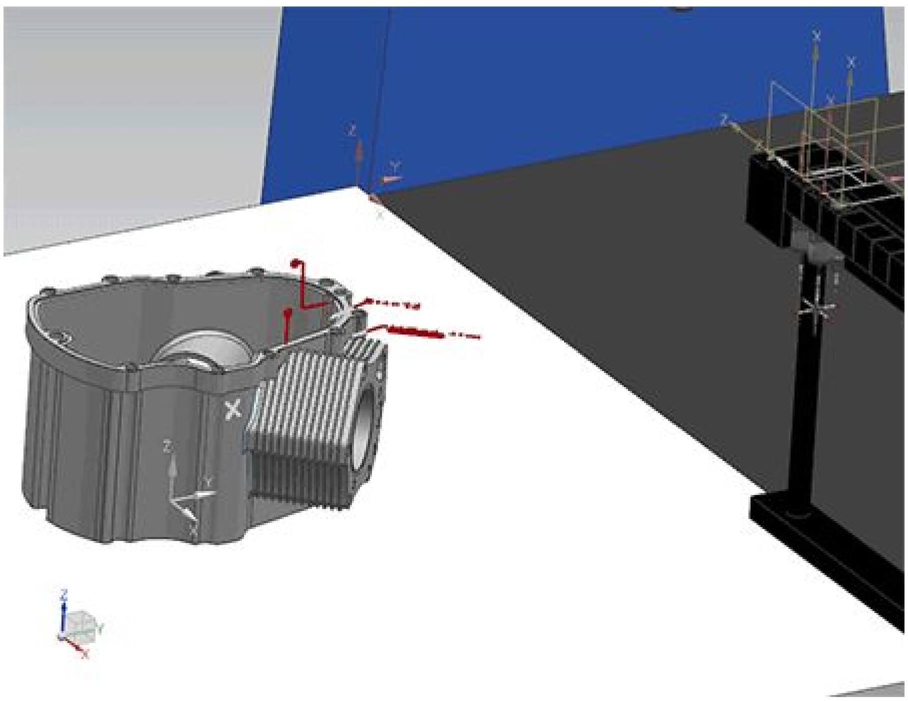
Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection file with no virtual machine loaded.
Inspection Navigator	In **Machine View**, double-click **GENERIC_MACHINE** and load the components.

New tool changing rack capability

What is it?

You can now set up virtual machine and head tools to support virtual tool changing racks, such as the following virtual Auto Change Rack (ACR).



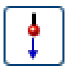
Why should I use it?

Changing racks extend the inspection capabilities of your CMM to include a variety of probes, extension lengths, and stylii. A rack can protect stored modules from environmental contaminants.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection file and load a virtual machine that contains a virtual changing rack. To use an actual changing rack you must configure your cmm execution software, or insert a CMM command such as the following DMIS statement into your program. `TH(Rack1)=THLDEF/SS(Tool1),1,SS(Tool2),2,SS(Tool3),3` Once set up, any probe change is automatically reflected in your simulations.

Inspection Path dialog box enhancements



Sub-operations list enhancements

What is it?

The Sub-Operations list in the **Inspection Path** dialog includes three new columns:

- The probe **Tip** index—0 for single-tipped probes, 0 or higher for multi-tip probes.
- If applicable, the probe’s **A Angle**, in degrees. The A angle usually rotates the probe barrel around the X-axis.
- If applicable, the probe’s **B Angle**, in degrees. The B angle usually rotates the head around the Z-axis.

Why should I use it?

The probe tip and angles reported for each point or sub-operation let you quickly view the sensor strategies used for each point and sub-operation in the inspection path. You can select multiple points and sub-operations in the Sub-Operations list and then apply tip, probe, and angle settings globally to all those selected points and sub-operations.

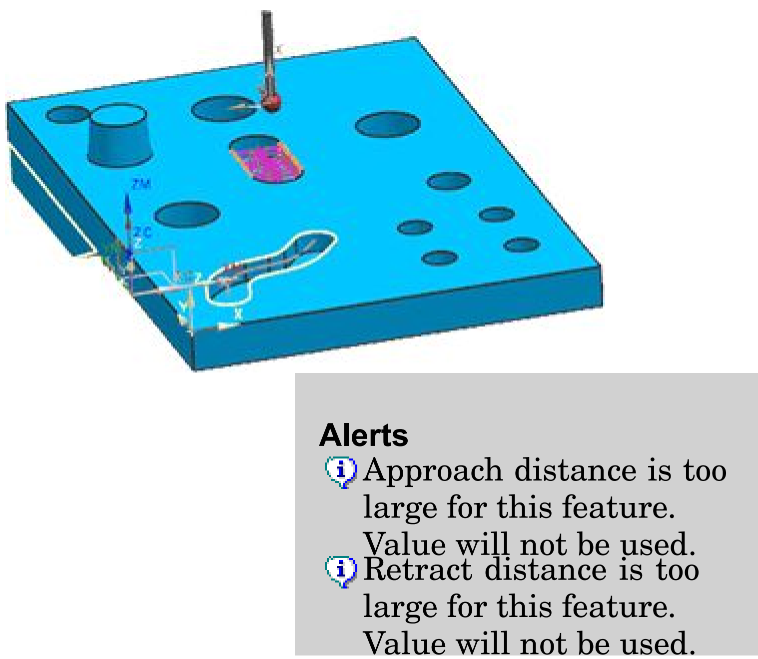
Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection file, and any necessary inspection features in PMI or CMM Inspection Programming.
Toolbar	**Insert® Inspection Path**
Menu	**Insert® Inspection Path**
Inspection Navigator	Right-click any node and choose **Insert® Inspection Path**

Enhanced approach logic and warnings

What is it?

The default programming logic related to how probes approach and retract from small holes and slots is enhanced. When creating inspection paths and sub-operations on such features, alerts will report potential problems in your process, and require you to fix those problems before committing your path.



Why should I use it?

Path warnings help you correct problems when you create an inspection path, rather than encountering them later during simulations.

Where do I find it?

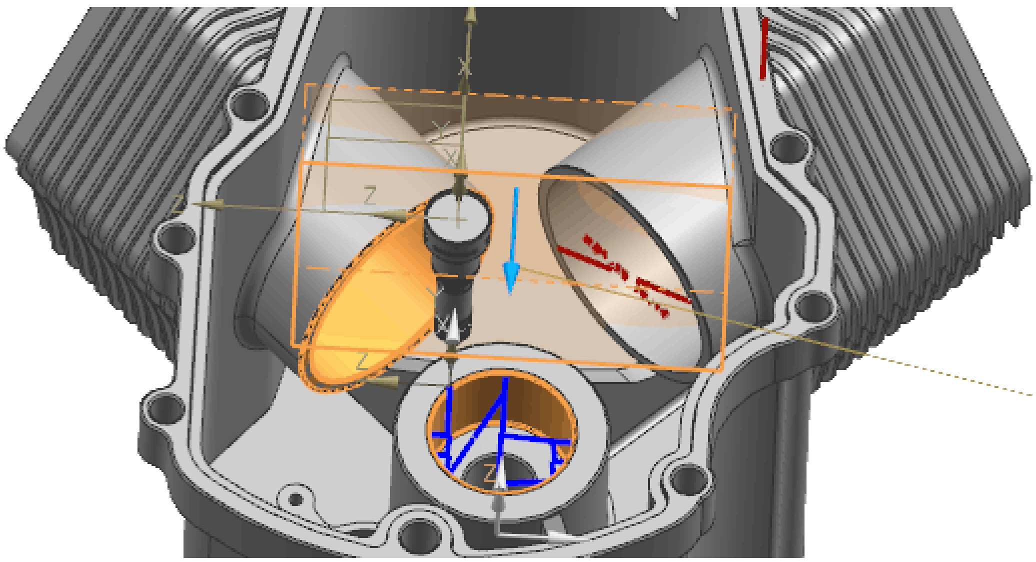
Application	CMM Inspection Programming
Prerequisite	You must create an inspection file, necessary inspection features, and an inspection path on a small hole or slot.

Safe plane improvements

What is it?

When you define an inspection path and then click **Specify Safe Plane** , the standard NX **Plane** dialog box now provides many ways to define a safe plane from part and machine geometry, as well as based on angles, distance, and mathematical coefficients. In some cases the safe plane definition is saved to the work part as a WAVE link so that the relationship between the inspection part file and the work part is recorded.

This example shows a safe plane created by bisecting the planes from the planar surfaces of the two cylinders adjacent to the cylinder being inspected. The safe plane is offset to keep the probe from colliding with the adjacent cylinders.

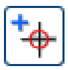


Show Clearance Plane also  displays a better representation of the path's actual safe plane than in previous releases.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection file, necessary inspection features, and an inspection path.
Inspection Path dialog box	**Transitions** tab® **Transition Points** group® • **Specify Safe Plane**  • **Show Clearance Plane** 

Inspection path sub-operation enhancements



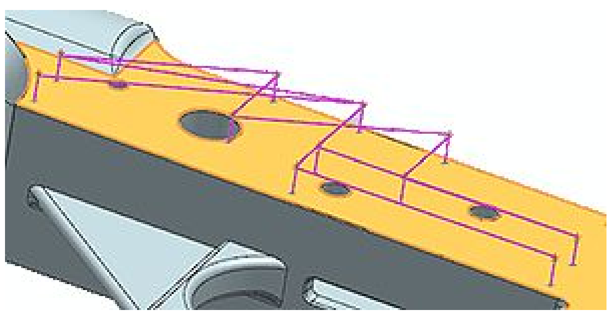
General sub-operation enhancements

What is it?

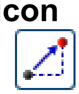
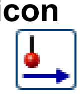
An inspection path can contain sub-operations of many different types, each of which depends on the type of feature being inspected. NX 8.0 Beta includes the following sub-operation enhancements.

- You can quickly create multiple **Measured Point** sub-operations from the **Create Sub-Operation** dialog box by clicking **Apply** after selecting each point location.

- A magenta inspection path preview now displays when you edit any parameter in a point set sub-operation except sensors. When you complete the sub-operation, the preview path is replaced by the standard blue lines to indicate the completed point set path.

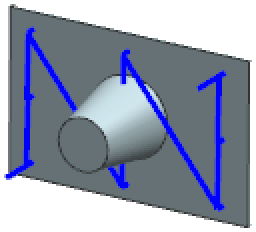


- Icons for the following sub-operation types have been changed to better represent their CMM functionality.

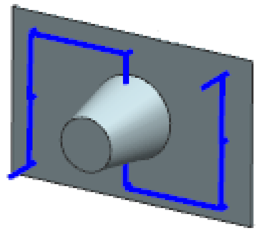
	Old icon	New icon		Old icon	New icon		Old icon	New icon
Scan Curve			**Scan Line**			**Scan Arc**		

- When you create point set sub-operations on a plane, cylinder, surface, sphere, cone, or torus inspection feature, you can select the sequence and U/V direction in which the measurement points are inspected.

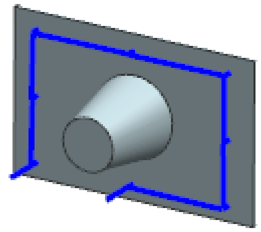
  Zig causes the probe to zig unilaterally across the grid, one line at a time.

  Zig Zag causes the probe to zigzag back and forth across each line in the grid.

  Nearest causes the probe to move from each point to its nearest neighbor.

Where do I find it?

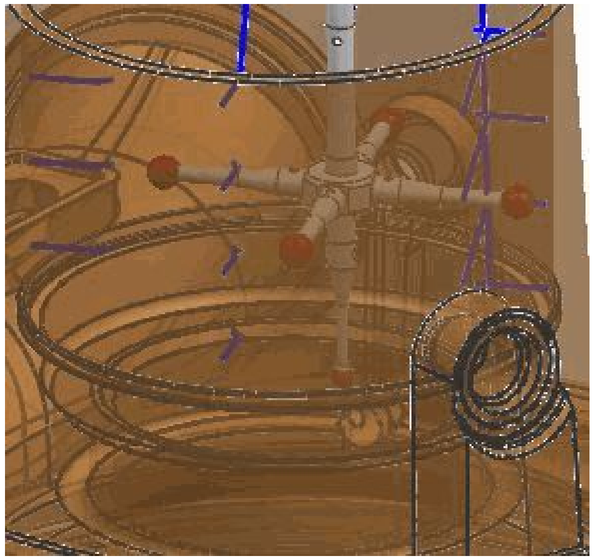
Application	CMM Inspection Programming
Prerequisite	You must create an inspection file, necessary inspection features in PMI or CMM Inspection Programming, and an inspection path.
Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation** ® **Type** group® sub-operation type

Custom sensor options in sub-operations

What is it?

You can either apply your own custom sensor to an inspection path, or you can let the application automatically create and apply a new sensor as needed. When you create or edit measurement points, transition points, and sub-operations from the **Inspection Path** dialog box, a wide variety of options related to sensor selection, path approach strategy, tips, and angles are available.

This example shows a multi-tip probe inspecting a cylinder using a point grid sub-operation. For each point location, the user has applied the most efficient star probe tip using sensor options.

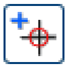


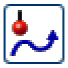
Why should I use it?

Sensor options let you:

- Specify whether the application should automatically create a new sensor as needed, or select the best existing sensor.
- Specify the sensor to be used.
- Manually specify probe angles.
- For multi-tip probes, specify a probe tip for the point or sub-operation.
- Specify probe angles as well as a specific probe tip.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must have: 1. An inspection file with at least one probe tool loaded. 2. A custom sensor, unless you want the application to automatically create one. 3. An inspection feature. 4. An inspection path on the inspection feature.
Inspection Path dialog box	**Measurement tab**® **Sub-Operations** group® • **Add Sub-Operation**  • **Name** list® double-click point or sub-operation

 **Scan sub-operation enhancements**

What is it?

One parameter has been modified, and three new parameters have been added, to both scan curve and 5-Axis scan curve sub-operations.

Parameters	
Minimum Points	Formerly **Number of Points**, this parameter sets the minimum number of points to be scanned. This number may increase as the **Curvature Factor** increases.
 **Minimum Spacing**	Sets the minimum space that must exist between measurement points on the curve.
 **Maximum Spacing**	Sets the maximum space that must exist between measurement points on the curve.
 **Curvature Factor**	Sets the level of point density at areas that represent the greatest bends in the curve. A factor of 0 sets no increase in point density due to curvature. The greater the factor, the more point density increases at areas of greater curvature.

Why should I use it?

Increasing the curvature factor, as well as setting minimum and maximum spacing between points, lets you better retrieve measurement data from the most significant and variable areas of a curve. These settings retrieve less data from straighter, less variable areas, and produce more meaningful statistical results.

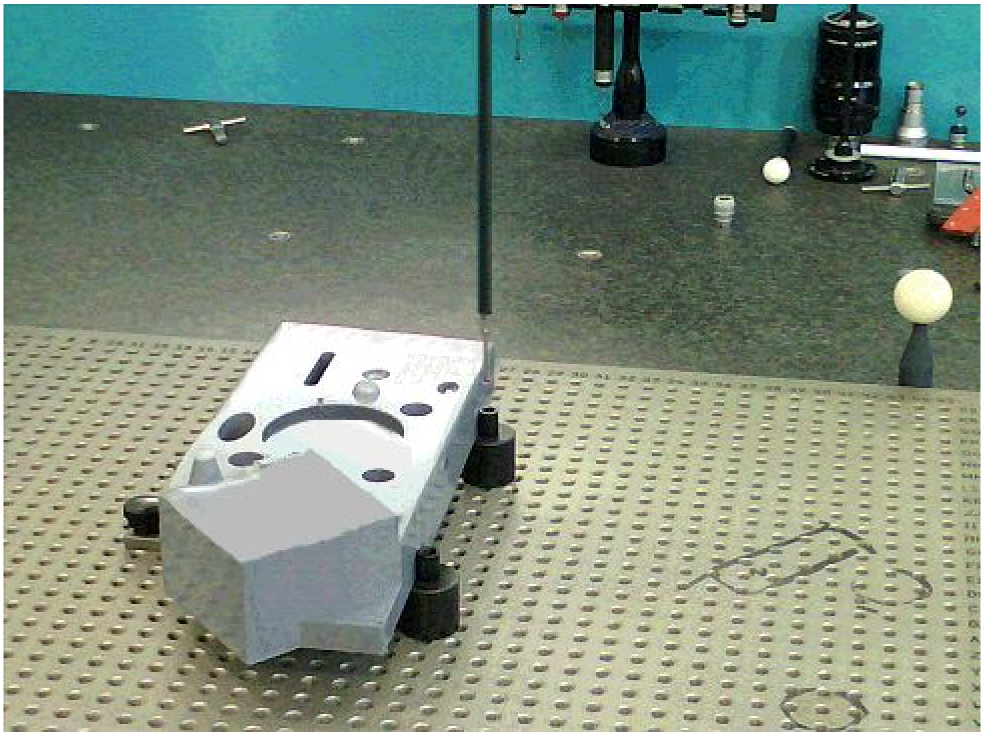
Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must open a work part file with curved geometry, create an inspection file, and create a curve inspection feature.
Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation** ® **Type** group® **Scan Curve** or **5 Axis Scan Curve**

5 Axis Scan Curve sub-operation

What is it?

Use a 5-axis scan curve sub-operation to inspect tight, irregular, or difficult-to-inspect areas of a complex model. Because 5-axis inspection systems quickly calibrate all angles that the probe can possibly use, they are generally preferable to three-axis scanners that require calibration each time an angle changes.



Why should I use it?

Use the synchronized motion of 5-axis heads and controllers to better reach difficult-to-inspect irregular features such as blades and blisks, and to reduce set up, programming, and measurement time.

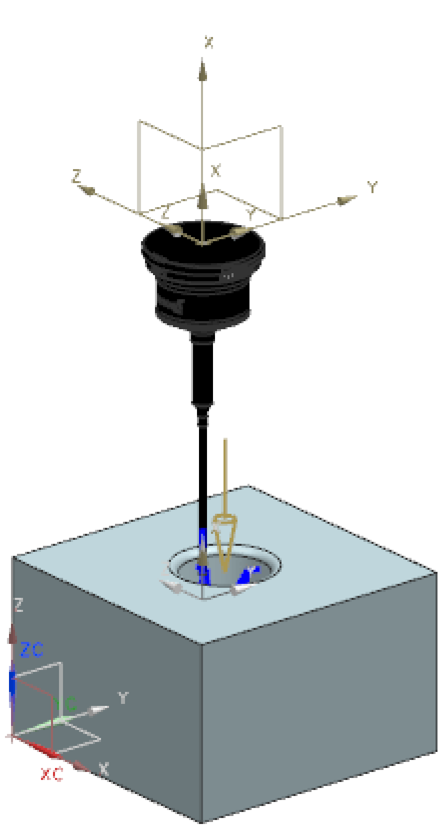
Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must open a work part file with geometry that allows you to create a curve inspection feature, create an inspection file, and load a virtual machine with a 5-axis head and probe.
Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation** ® **Type** group® **5 Axis Scan Curve**

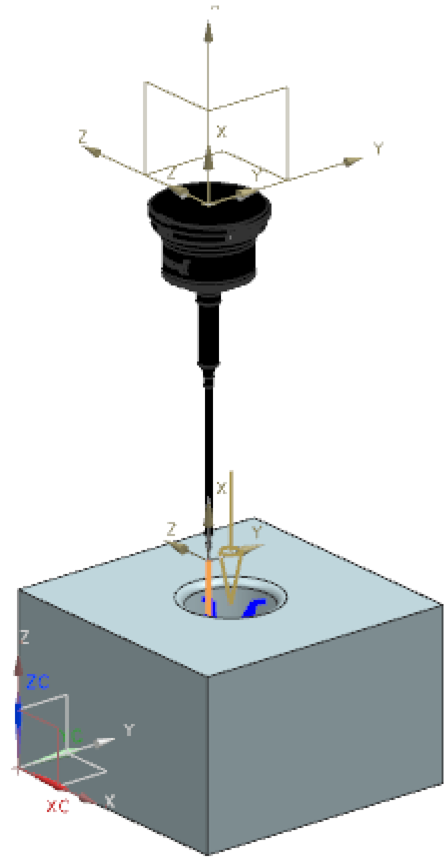
Delta Move sub-operation

What is it?

Use a **Delta Move** sub-operation to move your probe incrementally from its current location to another location, using either Work Coordinate System (WCS) or Machine Coordinate System (MCS) coordinates.



Probe at the stop location of an inspection path on a cylinder feature, with the MCS visible above the probe and WCS visible to the left of the work part



Probe at the location of a delta move sub-operation based on the WCS, with **Delta ZC** set to **40** so that the probe moves 40 millimeters from its previous location

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must open a work part file and create an inspection file.
Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation** ® **Type** group® **Delta Move**

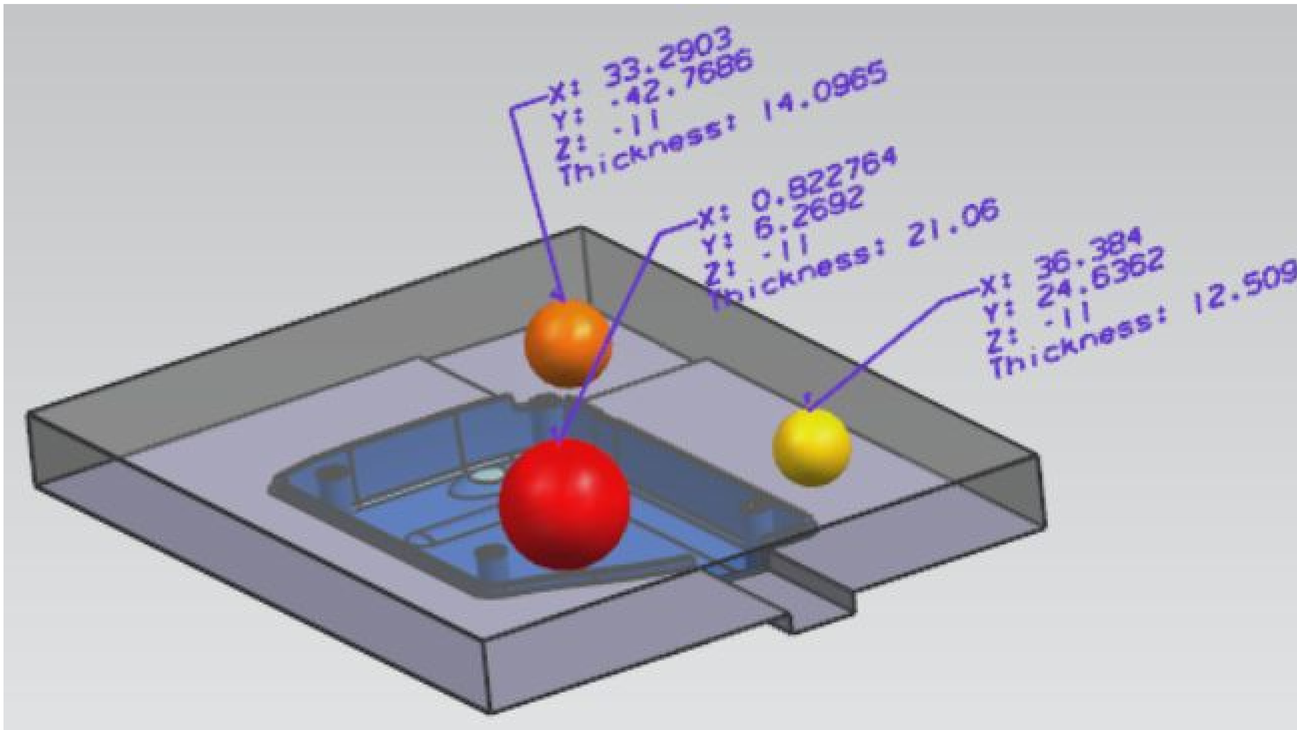
Molded Part Validation enhancements

What is it?

Molded Part Validation has been redesigned to be more intuitive and easy to use. **Check Regions** and **Check Wall Thickness** are now two separate commands. When you use either of these commands, you can:

- Use the **Select** option in the dialog box to select a body for analysis.
- Change your selection without closing the dialog box.
- Record and playback your analysis by using journaling.

The wall thickness check in **Molded Part Validation** now includes a dynamic results display, in which spheres of different colors and sizes represent thickness at the points that you select. You can also display PMI labels for thickness results.



Where do I find it?

Application	Mold Wizard
Menu	**Analysis→Molded Part Validation→Check Regions** or **Check Wall Thickness**

Chapter

9 Tooling Design

Die Engineering

Addendum Section enhancements

What is it?

It is now easier to use the **Reuse Library** from the **Addendum Section** dialog box.

- Thumbnails of the addendum section templates are available in the **Member View** subgroup of the **Section** group.
- The **Search** and **Folder View** subgroups of the **Section** group are removed.
- Addendum section templates for metric parts are saved in *\ugautomotive\diedes\reuse_lib\metric\addendum_section* and addendum section templates for English parts are saved in *\ugautomotive\diedes\reuse_lib\english\addendum_section*.

Where do I find it?

Application	Modeling
Prerequisite	The **Section** group is available only when you select a face in the **Product** group. The **Section Shape** list is available only when you select **At Point** or **At Plane** in the **Section** group.
Toolbar	**Die Engineering® Addendum Section**
Menu	**Tools® Vehicle Manufacturing Automation® Die Engineering® Addendum Section**
Location in dialog box	**Section** group® **Section Shape** list® **Reuse Library**

Edit Addendum Section

What is it?

Use the **Edit Addendum Section** command to open the **Section Parameters** dialog box and edit the parameters for a selected addendum section.

Note Currently, the **Edit Addendum Section** command works only after you access any command on the **Die Engineering** toolbar in the Modeling application.

Where do I find it?

Application	Modeling
Graphics window	Right-click an addendum section feature® **Edit Addendum Section**
Part Navigator	Right-click an addendum section® **Edit Addendum Section**

Pierce Task enhancements

What is it?

When you use the **Pierce Task** command, you can now do the following.

- You can view the press direction line of a curve as you select the curve for piercing. The value of the angle between the press direction line of the curve and the normal direction of the plane on which the curve lies is also displayed.
- You can detect whether the press direction of the pierce equipment is safe by checking the color of the press direction line of a curve.
    - Green = Safe
    - Red = Unsafe
    - Yellow = Needs investigation

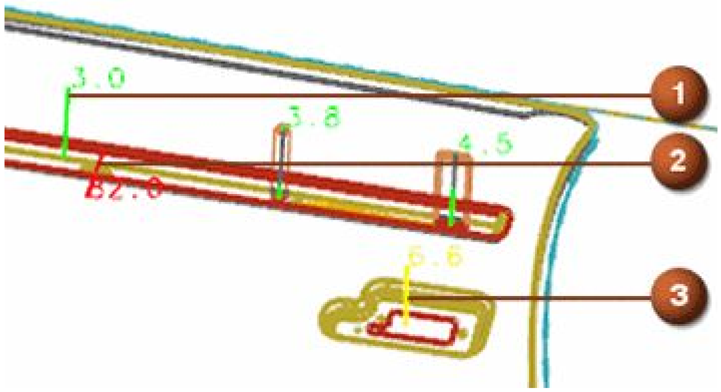
  The color of the line is determined by the values you set for the following new options in the **Pierce Task** dialog box.

  Angle Check Lower Limit If the angle between the press direction of a curve and the normal direction of the plane on which the curve lies is below this value, the color of the line is green.

  Angle Check Upper Limit If the angle between the press direction of a curve and the normal direction of the plane on which the curve lies is above this value, the color of the line is red.

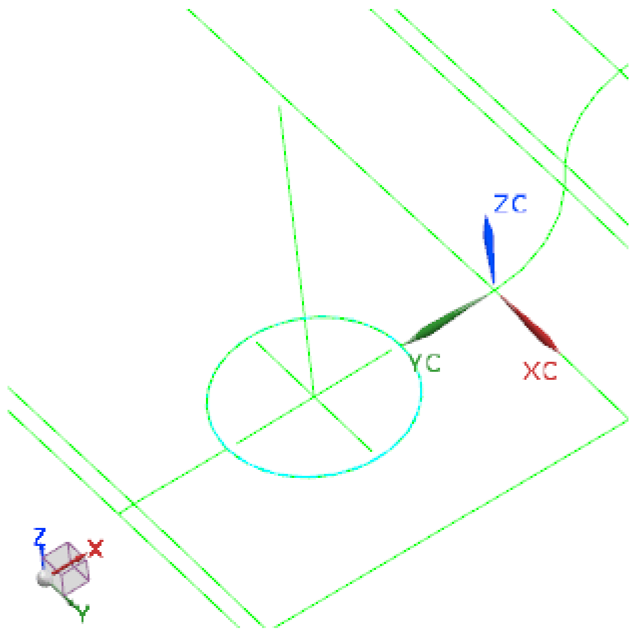
Note If the angle between the press direction of a curve and the normal direction of the plane on which the curve lies is in between the two values, the color of the line is yellow.

In the example, **Angle Check Lower Limit** is set to 5 and **Angle Check Upper Limit** is set to 10.

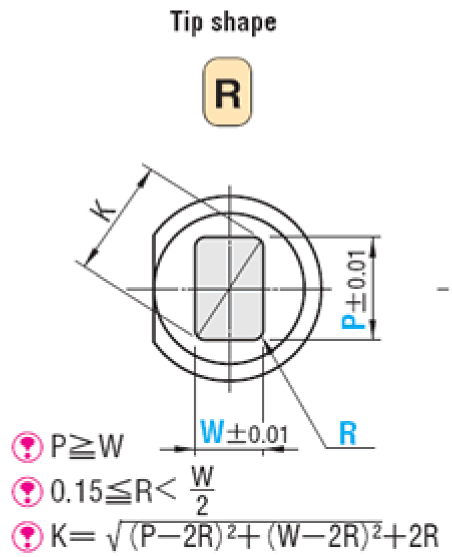


1	Press direction is safe
2	Press direction is unsafe
3	Press direction needs investigation

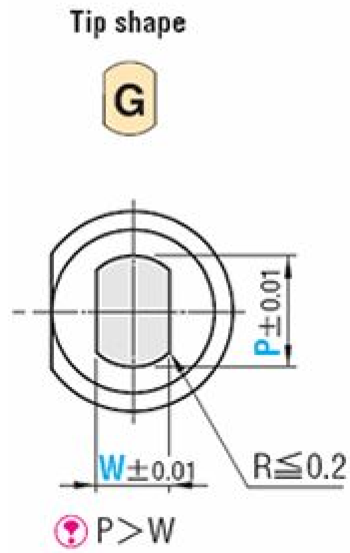
- You can view the center lines and the press direction line of a pierced hole.



- You can pierce the following new types of holes.



Rounded rectangular



Chord rectangular

Why should I use it?

In previous releases, to check the angle between the press direction of a curve and the normal direction of the plane on which the curve lies, you needed to close the **Pierce Task** dialog box and use the **Measure Angle** command.

Where do I find it?

Application	Modeling
Toolbar	**Die Engineering® Pierce Task** 
Menu	**Tools® Vehicle Manufacturing Automation® Die Engineering® Pierce Task**
Location in dialog box	**Settings** group® **Angle Check Upper Limit** or **Angle Check Lower Limit**

Trim Angle Check enhancements

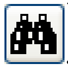
What is it?

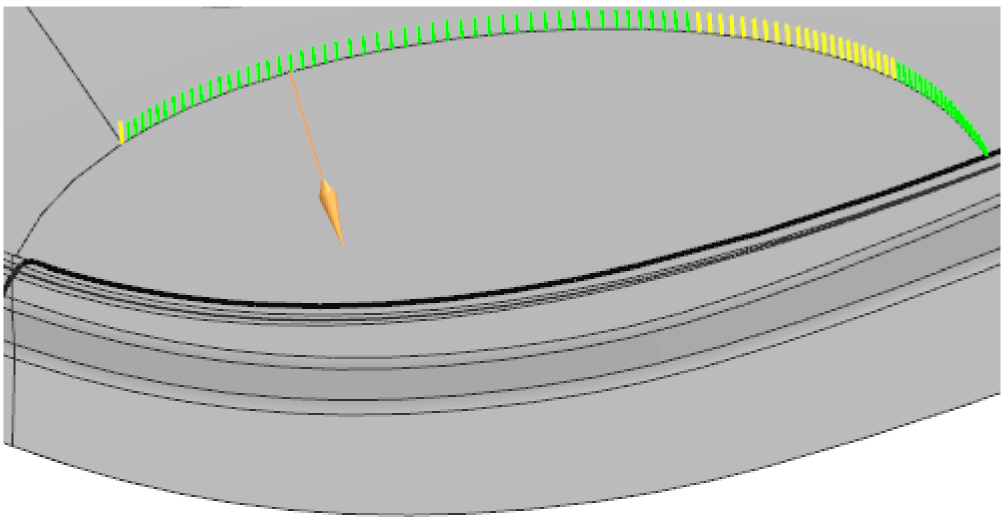
The **Trim Angle Check** command is enhanced to also display the output in the form of line segments of variable lengths and colors in the graphics window.

The length of a line segment indicates the trim angle at a sample check point and the color indicates whether the trim angle is compliant with manufacturing specifications.

- Green = Compliant
- Red = Noncompliant
- Yellow = Within a specified range of becoming noncompliant

Note You can specify the range in the **Dangerous Zone Range** customer default.

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



Why should I use it?

You can now check if a trim angle is compliant with manufacturing specifications by checking the color and the length of the line segments in downstream piercing punch or trimming design.

Where do I find it?

Application	Modeling
Toolbar	**Die Engineering® Trim Angle Check** 
Menu	**Tools® Vehicle Manufacturing Automation® Die Engineering® Trim Angle Check**

Die Design

Draw Punch

What is it?

Use this command to create a draw punch which forms the shape in the sheet metal. This command is similar to the **Draw Die Punch** command but it has additional options and enhanced performance.

Note The **Draw Die Punch** command is now called **Draw Die Punch (Legacy)**.

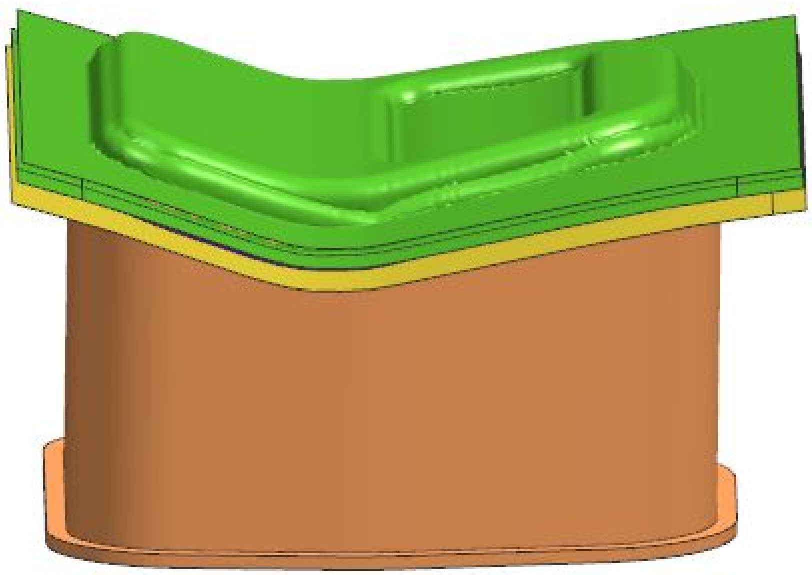
When you use the **Draw Punch** command you can:

- Use the standard and reusable templates from the **Reuse Library** to add details such as a rib, a keyway, or a handling hole. You can also define your own templates and then add them to the **Reuse Library**.

- Identify a failed feature and repair it.

- Create the core draw punch using the **Core Draw Punch** option.

- Use any one of the two methods, exact and rough offset, to create the draw punch. If the sheet body from Die Engineering or other CAD systems is imperfect, or has many gaps and fragments, you can have NX create a rough offset feature based on the input body and you can trim the body to form the punch face.



Sample Punch

Where do I find it?

Application	Modeling
Toolbar	**Die Design® Draw Punch**
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Draw Punch**

Draw Die

What is it?

Use this command to create a draw die which forms the shape in the sheet metal. This command is similar to the **Upper Draw Die** command but it has additional options and enhanced performance.

Note The **Upper Draw Die** command is now called **Upper Draw Die (Legacy)**.

When you use the **Draw Die** command you can:

- Use the standard and reusable templates from the **Reuse Library** to add details such as a rib, a keyway, and a handling hole. You can also define your own templates and then add them to the **Reuse Library**.

- Identify a failed feature and repair it.

- Create the core draw die using the **Core Draw Die** option.

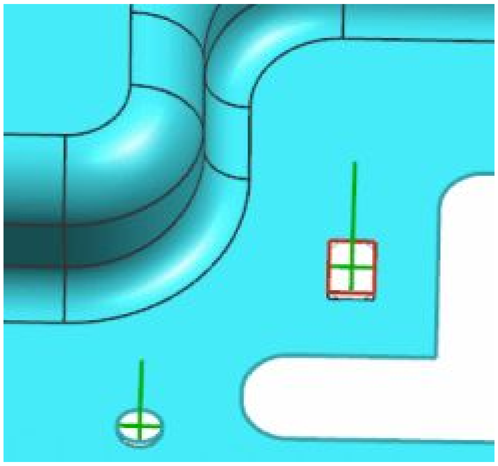
- Use anyone of the two methods, exact and rough offset, to create the draw die. If the sheet body from Die Engineering or other CAD systems is imperfect, or has many gaps and fragments, you can have NX create a rough offset feature based on the input body and you can trim the body to form the die face.

Where do I find it?

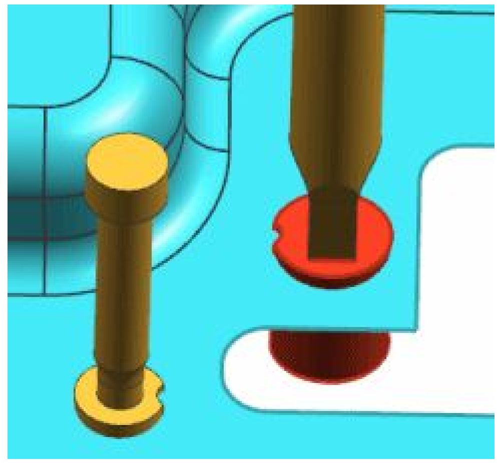
Application	Modeling
Toolbar	**Die Design® Draw Die**
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Draw Die**

Pierce Insert Design

Use the **Pierce Insert Design** command to select a hole curve created using the **Pierce Task** command and then insert a standard punch or die from the standard parts library.



Selected curves



Inserted punch and die

Note NX derives the pierce hole curve attributes from the **Pierce Task** command. Therefore, the **Pierce Insert Design** command works only for the hole curves that you created using the **Pierce Task** command.

Where do I find it?

Application	Modeling
Toolbar	**Die Design® Pierce Insert Design** 
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Pierce Insert Design**

Manufacturing Geometry enhancements

What is it?

When you use the **Manufacturing Geometry** command, you can now add or delete attributes of different objects that are included in the grouped faces.

In previous releases, you could view these attributes but you could not add or delete them.

Where do I find it?

Application	Modeling, Electrode Design, Mold Wizard, Progressive Die Wizard
Toolbar	(Modeling) **Die Design® Manufacturing Geometry** (Electrode Design) **Electrode Design® Manufacturing Geometry** (Mold Wizard) **Mold Wizard® Mold Tools® Manufacturing Geometry** (Progressive Die Wizard) **Progressive Die Tools® Manufacturing Geometry**
Menu	(Modeling) **Tools® Vehicle Manufacturing Automation® Die Design® Manufacturing Geometry** (Electrode Design) **Tools® Process Specific® Electrode Design® Manufacturing Geometry** (Mold Wizard) **Tools® Process Specific® Mold Wizard® Mold Tools® Manufacturing Geometry** (Progressive Die Wizard) **Tools® Process Specific® Progressive Die Tools® Manufacturing Geometry**

Reuse of Standard Parts

What is it?

The **Standard Part Library** in Die Design is now available through the **Reuse Library**. You can drag and drop standard parts into your assembly, and more easily customize standard parts to suit the needs of your design process.

The user interface of the **Standard Part Management** dialog box is redesigned to make the workflow easier and more intuitive.

Where do I find it?

Application	Die Design
Resource Bar	**Reuse Library** ® **Stamping Die Standard Part Library**
Toolbar	**Die Design® Standard Parts**
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Standard Parts**

Reuse of Standard Features

What is it?

The **Stamping Die Reusable Feature** in Die Design is now available through the **Reuse Library**. You can drag and drop reusable features into your draw punch and draw die, and you can add more features to suit the needs of your design process.

Where do I find it?

Application	Die Design
Resource Bar	**Reuse Library** ® **Stamping Die Reusable Feature**

Mold Wizard

Pocket enhancements

What is it?

The **Pocket** command now supports all thread types defined in NX. In addition, you can view the results of a pocket interference check in the HD3D environment.

Why should I use it?

The improvements to the **Pocket** command let you use many more threaded parts and make pocket checking easier to use and interpret.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard→Pocket** 
Menu	**Tools→Process Specific→Mold Wizard→Pocket**

Design Inserts enhancement

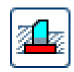
What is it?

When you import a design insert, you can now specify the parent. Previously, the relationship of the design insert to the solid body was hard-coded and you could not change it.

Why should I use it?

The ability to specify a parent part for a design insert gives you more flexibility in structuring your mold design assembly.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard→Mold Tools→Design Inserts** 
Menu	**Tools→Process Specific→Mold Wizard→Mold Tools→Design Inserts**

Stock Size enhancements

What is it?

When you use the **Stock Size** command, you can change the CSYS direction dynamically. You can specify a clearance, and you can create an expression attribute for stock size information.

Why should I use it?

The improvements to the **Stock Size** command let you position elements associatively, and customize clearance according to your design requirements. You can:

- Predefine a named CSYS as a default direction in part templates.
- Predefine stock size types and clearance in part templates.
- Generate stock size attributes by predefined CSYS type and clearance if you select multiple components in the bill of materials.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard→Mold Tools→Stock Size**
Menu	**Tools→Process Specific→Mold Wizard→Mold Tools→Stock Size**

Design Parting Surface enhancement

What is it?

When you use the **Extrude** method to create a parting surface, you can now specify a draft angle by using the new **Draft** option.

Why should I use it?

You can create a drafted parting surface within Mold Wizard. You do not have to go into the Modeling application to create the drafted parting surface.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard→Mold Parting Tools→Design Parting Surface**
Menu	**Tools→Process Specific→Mold Wizard→Parting Tools→Design Parting Surface**
Location in dialog box	**Create Parting Surface** group® **Method** subgroup® **Extrude**

Mold Design Validation enhancements

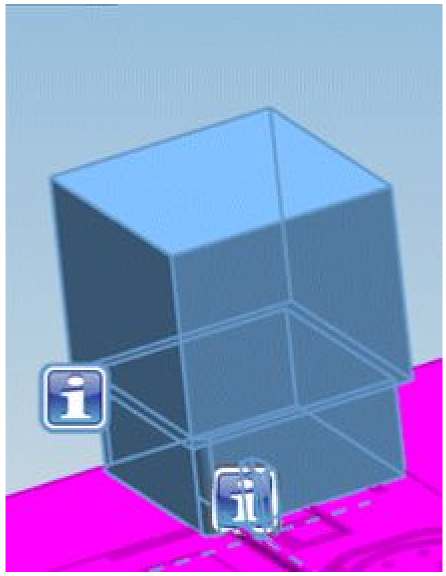
What is it?

Additional checkers are available from the **Mold Design Validation** dialog box, and the dialog box is redesigned to make it easier to use.

Mold Part Quality combines the **Undercut Checker** and **Draft Angle Checker**, which were previously available as separate checkers.

Component Validation checks are available for checking interferences and overlaps.

- **Electrode** checks for interferences in the electrode assembly.



- **Core/Cavity** validates the division of the part into cavity and core, checks the geometry of each sheet that is going to be sewn, and checks for and highlights overlapping sheets.

- **Pocket** checks for intersecting components, and highlights and shows all standard parts that have no corresponding pockets. (This checker is also available in the **Pocket** dialog box.)

Parting Validation has the following new checkers:

- **Overlap Patch Surface** checks for patch surfaces that overlap each other and must be repaired before you can create trim sheets.

- **Sheet Body Boundary** checks the boundaries of sheet bodies, and is especially useful for finding gaps in sheet bodies that patch internal loops.

Why should I use it?

The new checkers help you to ensure the manufacturability of your part.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard→Mold Design Validation**
Menu	**Tools→Process Specific→Mold Wizard→Mold Design Validation**

Reuse of standard parts

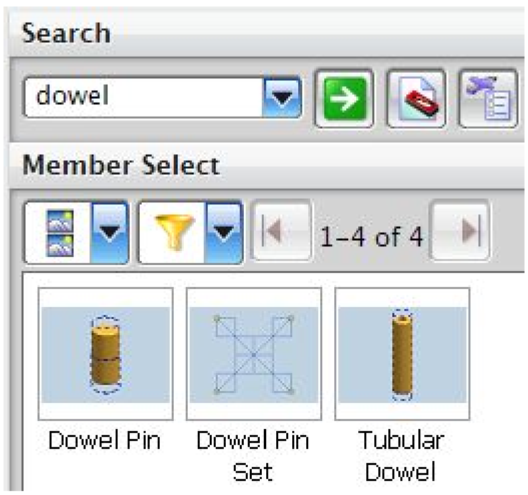
What is it?

The **Standard Part Library** in Mold Wizard is now available through the **Reuse Library** . You can drag and drop standard parts into your assembly, create families of mold components, and more easily customize standard parts to suit the needs of your design process.

The user interface of the **Standard Part Management** dialog box is redesigned to make the workflow easier and more intuitive.

- It now supports multiple standard part libraries.

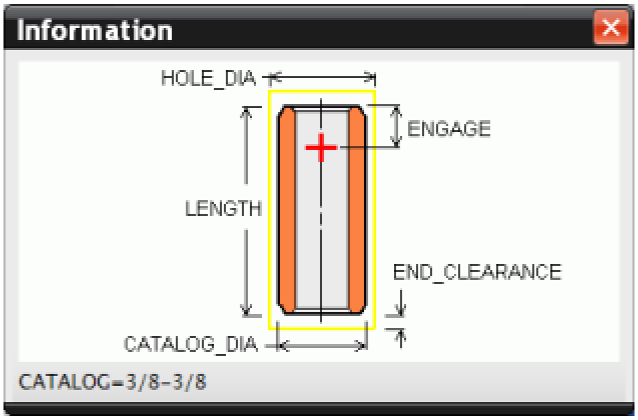
- You can search for a standard part in Mold Wizard standard part libraries.



- You can configure you own libraries separate from the default Mold Wizard library.

- You can control catalog visibility, for example, you can hide catalogs that you do not use.

- Bitmap, catalog information, and comments appear in a separate Information window.



Why should I use it?

Accessing standard parts through the **Reuse Library** simplifies the **Standard Part Management** dialog box, making design information clearer and easier to use. It gives you access to more standard part libraries and makes it easier to find the parts you are looking for.

Where do I find it?

Standard Part Management

Application	Mold Wizard
Resource Bar	**Reuse Library** →**MW Standard Part Library**
Toolbar	**Mold Wizard→Standard Part Library**
Menu	**Tools→Process Specific→Mold Wizard→Standard Part Library**

Standard Part Library Management

Resource Bar	**Reuse Library** →**MB3 on Name column→Library Management**
Toolbar	**Reuse Library** ® **Reuse Library Management**

Mold Wizard library enhancements

What is it?

The **Standard Part Library**, the **Slide and Lifter Library**, the **Cooling Standard Part Library**, and the **Insert Library** are all improved in the following ways:

- The metric and English catalogs are now up-to-date.
- New components are added.
- Retired components are removed.
- Many new vendor libraries are available.

Why should I use it?

The libraries provide more up-to-date selections of parts.

Where do I find it?

Application	Mold Wizard
Resource Bar	**Reuse Library** →**Mold Wizard Standard Part Library**
Toolbar	**Mold Wizard**→**Standard Part Library** or **Slide and Lifter Library** or **Cooling Standard Part Library** or **Insert Library**
Menu	**Tools**→**Process Specific**→**Mold Wizard**→**Standard Part Library** or **Slide and Lifter Library** or **Insert Library** **Tools**→**Process Specific**→**Mold Wizard**→**Cooling Tools**→**Cooling Standard Part Library**

Cooling enhancements

What is it?

Mold Cooling Tools enhancements include:

- A **Cooling Circuits** command that you can use to create and analyze water flow to help you complete your cooling circuit design. When you use this command, you can define a circuit that needs to be marked, get related fittings, set the color for the circuit bodies, and classify the different circuits by name.

- A **Define Channel** command that lets you recognize existing channel bodies or cylinder bodies as Mold Wizard cooling channels and cooling baffles, lets you change their color and layer, and lets you add COOLING_CHANNEL attributes to the bodies for cooling channels or COOLING_CHANNEL and COOLING_BAFFLE attributes to the bodies for cooling baffles.

- Changes to the **Cooling Fittings** command that let you add a cooling baffle automatically and select an intersection body so that fittings can be designed in one part file. You can also have baffles created automatically when the cooling body has both COOLING_CHANNEL and COOLING_BAFFLE attributes. In this case, NX determines the baffle length.

Why should I use it?

You can design cooling fittings and cooling circuits quickly and easily.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard→Mold Cooling Tools**
Menu	**Tools→Process Specific→Mold Wizard→Cooling Tools**

Create Box enhancements

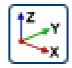
What is it?

The **Create Box** dialog box has been redesigned so that the order in which you enter your inputs now matches your workflow. Other enhancements to the **Create Box** command include the following.

- You can specify the orientation of the box.

- You can specify the color of the box.

Where do I find it?

Application	Mold Wizard, Progressive Die Wizard, Electrode Design
Toolbar	**Mold Tools** or **Progressive Die Tools** or **Electrode Design® Create Box** 
Menu	**Tools® Process Specific® Mold Wizard® Mold Tools® Create Box** **Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® Create Box** **Tools® Process Specific® Electrode Design® Create Box**
Location in dialog box	**Reference CSYS® Specify Orientation**  **Settings® Color**

Replace Solid enhancements

What is it?

The **Replace Solid** dialog box has been redesigned so that the order in which you enter your inputs now matches your workflow. Other enhancements to the **Replace Solid** command include the following.

- A **Reverse Direction** option that lets you reverse the direction of the replacement solid.
- A **Remove Parameters** check box that, when you select it, removes feature parameters from the bounding box.
- A **Clearance** option that lets you change the clearance value.

Where do I find it?

Reverse Direction

Application	Mold Wizard, Progressive Die Wizard, Electrode Design
Toolbar	**Mold Tools** or **Progressive Die Tools** or **Electrode Design® Replace Solid**
Menu	**Tools® Process Specific® Mold Wizard® Mold Tools® Replace Solid** **Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® Replace Solid** **Tools® Process Specific® Electrode Design® Replace Solid**
Location in dialog box	**Replacement Face® Reverse Direction** **Settings® Remove Parameters** **Settings® Clearance**

Trim Solid enhancements

What is it?

The **Trim Solid** dialog box has been redesigned so that the order in which you enter your inputs now matches your workflow. Other enhancements to the **Trim Solid** command include the following.

- A **Select Target Component** option that lets you link the resulting trim body to a selected component.

- A **Remove Parameters** check box that, when you select it, removes feature parameters from the bounding box.

Where do I find it?

Application	Mold Wizard, Progressive Die Wizard, Electrode Design
Toolbar	**Mold Tools** or **Progressive Die Tools** or **Electrode Design® Trim Solid**
Menu	**Tools® Process Specific® Mold Wizard® Mold Tools® Trim Solid** **Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® Trim Solid** **Tools® Process Specific® Electrode Design® Trim Solid**
Location in dialog box	**Target® Select Target Component** **Settings® Remove Parameters**

Progressive Die Design

Quick Quotation

What is it?

Use the **Quick Quotation** command to calculate a quick cost estimation of your progressive die based on your conceptual design. You can estimate your complete design, or you can run an early estimate when you finish the strip layout.

The **Quick Quotation** dialog groups have the following uses.

- Use the **Job Information** options to specify the job information, such as the quote number and customer name.
- Use the **Project Definition** options to specify the base information of a project.
- Use the **Concept Design** options to make a conceptual design for the die base and each of the insert groups.
- Use the **Grouping** options to group the profile defined in **Concert Design** into a different insert group. For example, you can identify a selected insert group as piercing or bending.
- Use the **Setting** options to choose a material for the insert group and to configure the quick quotation template. You can change the template language, add materials, and change the material heat-treatment cost.

Why should I use it?

The **Quick Quotation** command provides an automated standard method to produce a cost estimation. Currently, most companies use manual methods that are time-consuming and unreliable because they are based on the experience of the quoter.

Where do I find it?

Application	Progressive Die Wizard
Prerequisite	You must initialize a progressive die project.
Toolbar	**Progressive Die Wizard® Quick Quotation**
Menu	**Tools® Process Specific® Progressive Die Wizard® Quick Quotation**

Changeover Management

What is it?

Use the **Changeover Management** command to reuse an existing die design for a similar sheet metal part. You can then modify the existing design for a new part, and also keep the previous design in one project assembly.

The **Changeover Management** command also helps you manage multiple changeovers for different sheet metal parts.

In the **Changeover Management** dialog box, the **Changeovers** tree lists all the changeovers in your current project assembly, with a green check beside the currently-used changeover. You can select a different changeover for use, or you can add or remove components from a selected changeover.

When you select a component in the graphics window, any changeovers that use it are displayed in the Status line. The bill of material (BOM) reports only the components related to the current active changeover.

Why should I use it?

When the die design you need is similar to an existing design, creating a changeover instead of starting the design from scratch can save significant time and effort. All unchanged designs can be shared for different product packages.

Where do I find it?

Application	Progressive Die Wizard
Prerequisite	You must initialize a progressive die project.
Toolbar	**Progressive Die Wizard® Workflow Management** ® **Changeover Management**
Menu	**Tools® Process Specific® Progressive Die Wizard® Workflow Management® Changeover Management**

Concurrent Design Management

What is it?

Use the **Concurrent Design Management** command when multiple designers need to work on a progressive die design project simultaneously. The project lead assigns the design tasks to the designers, and each designer works under a separate node.

Why should I use it?

The advantages of using the **Concurrent Design Management** command include the following.

- Having multiple designers work on the same project shortens the delivery time.
- Other designers can modify and save in their own node without causing unintentional modifications and saves in your node. If any of their changes affect your node, you can wait until a convenient time to update.
- Each designer can check and reference other designers' designs.
- You can use this command in both native NX and the Teamcenter Integration for NX environments.

Where do I find it?

Application	Progressive Die Wizard
Prerequisite	You must initialize a progressive die project.
Toolbar	**Progressive Die Wizard® Workflow Management** ® **Concurrent Design Management**
Menu	**Tools® Process Specific® Progressive Die Wizard® Workflow Management® Concurrent Design Management**

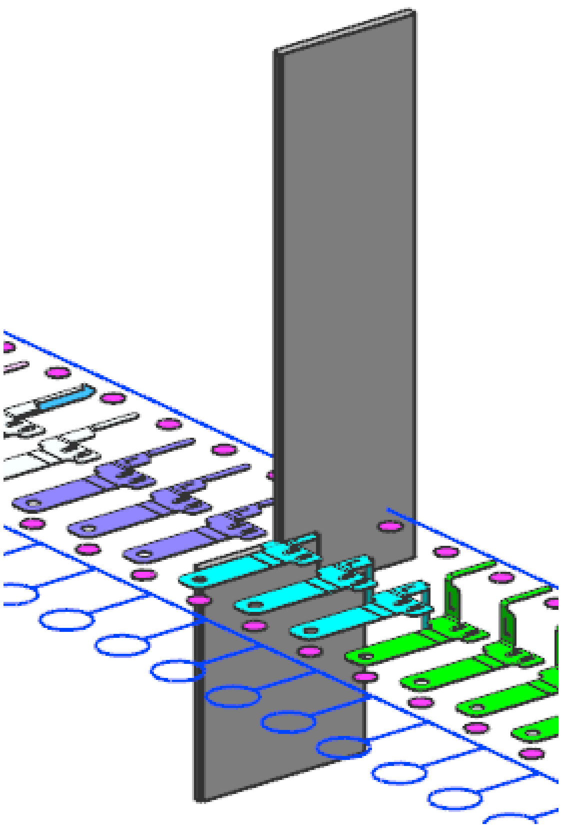
Bending Insert Design

What is it?

The **Bending Insert Design** command, which in NX 7.5 was a tab on the **Insert Group** dialog box, now has a separate toolbar button and dialog box. The **Bending Insert Design** dialog box has been redesigned so that the order in which you enter your inputs now matches your workflow.

Other enhancements include the following:

- You can classify the die and punch when you load a bending insert using the **Standard Parts** command.
- You can create a user-defined bending insert.



Where do I find it?

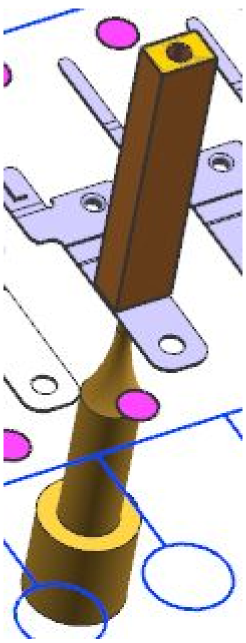
Application	Progressive Die Wizard and Engineering Die Wizard
Prerequisite	You must initialize a progressive or engineering die project.
Toolbar	**Progressive Die Wizard** or **Engineering Die Wizard**
Menu	**Tools® Process Specific® Progressive Die Wizard** or **Engineering Die Wizard® Bending Insert Design**

Burring Insert Design

What is it?

The **Burring Insert Design** command, which in NX 7.5 was a tab on the **Insert Group** dialog box, now has a separate toolbar button and dialog box. The **Burring Insert Design** dialog box has been redesigned so that the order in which you enter your inputs now matches your workflow.

You can rename burring punch or die components when you create them.



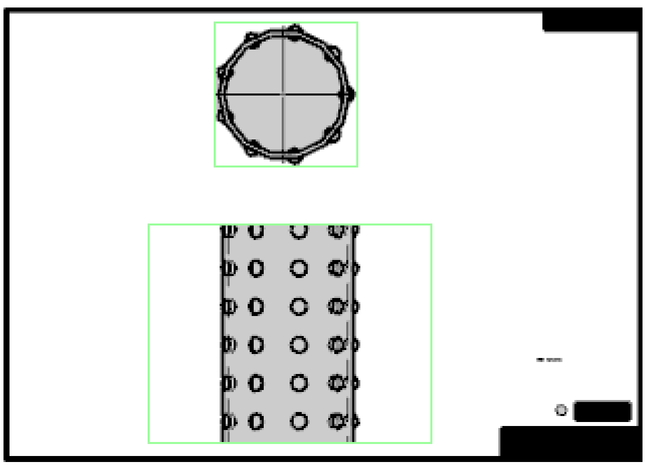
Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard
Prerequisite	You must initialize a progressive or engineering die project.
Toolbar	**Progressive Die Wizard** or **Engineering Die Wizard® Burring Insert Design**
Menu	**Tools® Process Specific® Progressive Die Wizard** or **Engineering Die Wizard® Burring Insert Design**

Component Drawing enhancements

What is it?

The **Component Drawing** command is enhanced to let you specify different templates, drawing names, or sheet names for component drawings before you create them. Also, you can now easily list the parts with existing drawings.



Why should I use it?

Many die and mold companies use component drawings to communicate design information to machinists. Mold and die designs may require hundreds of component drawings, so a well-designed automatic drawing tool that can create component drawings in batch is essential. Specifying dimensions and annotations for the drawings as part of their batch creation, instead of manually finishing each drawing after batch creation, can save you a lot of time and effort.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Prerequisite	You must enter the Drafting application.
Toolbar	**Mold Wizard** or **Progressive Die Wizard® Mold Drawing Drop-down** or **Progressive Die Drawing Drop-down** list ® **Component Drawing**
Menu	**Tools® Process Specific® Mold Wizard** or **Progressive Die Wizard® Mold Drawing** or **Progressive Die Drawing® Component Drawing**

Standard part catalogs

What is it?

The library used by the **Standard Parts** command is enhanced with an updated catalog of punches from MiSUMi, one of the largest standard part companies. The punches are used in progressive dies and other dies that you can create with the Progressive Die Wizard and the Engineering Die Wizard.

Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard
Toolbar	**Progressive Die Wizard** or **Engineering Die Wizard® Standard Parts**
Menu	**Tools® Process Specific® Progressive Die Wizard** or **Engineering Die Wizard® Standard Parts**

Progressive Die Wizard enhancements

What is it?

The following **Progressive Die Wizard** commands have been redesigned for easier use.

- **Die Base**
- **Bending Insert Design**
- **Burring Insert Design**
- **Clearance Management**

- **Corner Design**
- **Delete Files**

Why should I use it?

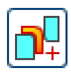
The redesigned dialog boxes are easier to use, because you fill them out from top to bottom, so your inputs match your workflow.

Where do I find it?

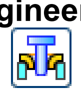
Die Base

Application	Progressive Die Wizard and Engineering Die Wizard
Toolbar	**Progressive Die Wizard** or **Engineering Die Wizard® Die Base** 
Menu	**Tools® Process Specific® Progressive Die Wizard** or **Engineering Die Wizard® Die Base**

Bending Insert Design

Application	Progressive Die Wizard and Engineering Die Wizard
Toolbar	**Progressive Die Wizard** or **Engineering Die Wizard® Bending Insert Design** 
Menu	**Tools® Process Specific® Progressive Die Wizard** or **Engineering Die Wizard® Bending Insert Design**

Burring Insert Design

Application	Progressive Die Wizard and Engineering Die Wizard
Toolbar	**Progressive Die Wizard** or **Engineering Die Wizard® Burring Insert Design** 
Menu	**Tools® Process Specific® Progressive Die Wizard** or **Engineering Die Wizard® Burring Insert Design**

Clearance Management

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Progressive Die Tools ® Clearance Management**
Menu	**Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® Clearance Management**

Corner Design

Application	Mold Wizard and Progressive Die Wizard
Toolbar	**Mold Wizard** or **Progressive Die Wizard® Mold Tools** or **Progressive Die Tools ® Corner Design**
Menu	**Tools® Process Specific® Mold Wizard** or **Progressive Die Wizard® Mold Tools** or **Progressive Die Tools® Corner Design**

Delete Files

Application	Mold Wizard and Progressive Die Wizard
Toolbar	**Mold Wizard® Unused Part Management** **Progressive Die Wizard® Progressive Die Tools® Delete Files**
Menu	**Tools® Process Specific® Mold Wizard® Unused Part Management** **Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® Delete Files**

Direct Unfolding enhancements

What is it?

The **Direct Unfolding** command is enhanced to create a top-level assembly only when you create intermediate stages.

Why should I use it?

When your project does not need intermediate stages, the number of part files that NX creates is reduced.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Sheet Metal Tools** ® **Direct Unfolding**
Menu	**Tools® Process Specific® Progressive Die Wizard® Sheet Metal Tools® Direct Unfolding**
Location in dialog box	**Type** group® **Create Intermediate Stages**

Analyze Formability — One-step enhancements

What is it?

The **Analyze Formability — One-step** command is enhanced as follows:

- For the **Entire Unform** type, you can now select either a solid body or faces. In previous releases, you could only select faces.

  When you select a solid body, the middle surface is automatically created.

- For the **Intermediate Unform** type, the **Outer Surface** and **Inner Surface** thickness options are now available. In previous releases, only the **Middle Surface** thickness option was available.

- For all types, the **Curve along Curve** option now supports all the curve options in selection intent. In previous releases, you could only select single curves.

Where do I find it?

Application	Progressive Die Wizard
Prerequisite	From the **Type** list, select **Entire Unform**.
Toolbar	**Progressive Die Wizard® Sheet Metal Tools** ® **Analyze Formability — One-step**
Menu	**Analysis® Analyze Formability — One-step**
Location in dialog box	**Object Type** group® **Object Type** list® **Solid**® select a solid body in the graphics window **Thickness** group® **Surface Type**® **Outer Surface/Inner Surface** **Boundary Conditions** group® **Constraint Type**® **Curve along Curve**

Reuse of standard parts

What is it?

The **Standard Part Library** in Progressive Die Wizard is now available through the **Reuse Library**. You can drag and drop standard parts into your assembly, and more easily customize standard parts to suit the needs of your design process.

The user interface of the **Standard Part Management** dialog box is redesigned to make the workflow easier and more intuitive.

Where do I find it?

Application	Progressive Die Wizard
Resource Bar	**Reuse Library** ® **PDW Standard Part Library**
Toolbar	**Progressive Die Wizard® Standard Part Library**
Menu	**Tools® Process Specific® Progressive Die Wizard® Standard Part Library**

Engineering Die Design (EDW)

Engineering Die Design

What is it?

Use the **Engineering Die Design** application to design various die types such as the following:

- Transfer die
- Tandem die
- Compound die
- Single die
- Blanking die
- Reverse die

The Engineering Die Design shares most of its functions with the Progressive Die Wizard. Items that are unique to the Engineering Die Design include the following.

- The **Station Management** command
- A die base for the Engineering Die Design for different die types

Why should I use it?

The Engineering Die Design helps you design a wide variety of die types.

Transfer dies are used for sheet metal parts without a carrier in their strip, such as covers for cell phones or video cameras.

Tandem dies are used for large sheet metal parts, such as automotive panels, TV covers, washing machines, or desktop computer cases.

Where do I find it?

Menu	**Start® All Applications® Engineering Die Design**

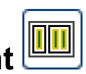
Station Management

What is it?

Use the **Station Management** command to define the engineering die station layout and station attributes, including the following.

- Create or delete a station.
- Define the station number and distance.
- Define the station name.
- Add or remove selected entities from the station.

Where do I find it?

Application	Engineering Die Design
Prerequisite	You must initialize an engineering die project. Choose **Tools**® **Process Specific**® **Engineering Die Design**® **Initialize Project**.
Toolbar	**Engineering Die Design**® **Station Management** 
Menu	**Tools**® **Process Specific**® **Engineering Die Design**® **Station Management**

Electrode Design

Electrode Fixture

What is it?

Use the **Electrode Fixture** command to do the following.

- Add electrode holder and pallet components from the Reuse Library to your electrode design assembly.
- Edit electrode holder and pallet components installed in your electrode design assembly.

Where do I find it?

Application	Electrode Design
Toolbar	**Electrode Design® Electrode Fixture**
Menu	**Tools® Process Specific® Electrode Design® Electrode Fixture**
Resource Bar	**Reuse Library** ® **Electrode Holder Library** or **Electrode Pallet Library**

Delete Body/Component

What is it?

Use the **Delete Body/Component** command to delete a selected electrode sparking body or component.

Where do I find it?

Application	Electrode Design
Toolbar	**Electrode Design® Delete Body/Component**
Menu	**Tools® Process Specific® Electrode Design® Delete Body/Component**

Check Electrode enhancements

What is it?

The **Check Electrode** command is enhanced to show the checking results in the HD3D **Check-Mate** dialog box.

The **Check Electrode** command replaces the **Electrode Checking** command that was available in NX 7.5.

Why should I use it?

Results are now kept with the electrode assembly. You can get the last checking status if you reopen the assembly.

Where do I find it?

Application	Electrode Design
Toolbar	**Electrode Design® Check Electrode**
Menu	**Tools® Process Specific® Electrode Design® Check Electrode**

Electrode Design enhancements

What is it?

The following **Electrode Design** commands are enhanced with redesigned dialog boxes.

- **Initialize Electrode Project**
- **Design Blank**
- **Electrode Drawing**

Enhancements to the **Design Blank** command include the following:

- Graphics are displayed in a separate window.
- Parameters of the blank are displayed in the **Design Blank** dialog box.
- The recommended sizes of the blank are displayed after you choose the electrode head bodies.

Enhancements to the **Electrode Drawing** dialog box include the following:

- You can generate EDM and CNC drawings.
- A4 size drawing sheet templates are available.
- You can change the drawing sheet template for existing drawings.
- Two drawing types, **Master Model** and **Self Contained**, are available.
- You can create PDF files for drawings.

Why should I use it?

The redesigned dialog boxes are easier to use, because you fill them out from top to bottom, so your inputs match your workflow.

Where do I find it?

Initialize Electrode Project

Application	Electrode Design
Toolbar	**Electrode Design® Initialize Electrode Project** or **Design Blank** or **Electrode Drawing** or **Check Electrode**
Menu	**Tools® Process Specific® Electrode Design® Initialize Electrode Project** or **Design Blank** or **Electrode Drawing** or **Check Electrode**

Weld Assistant

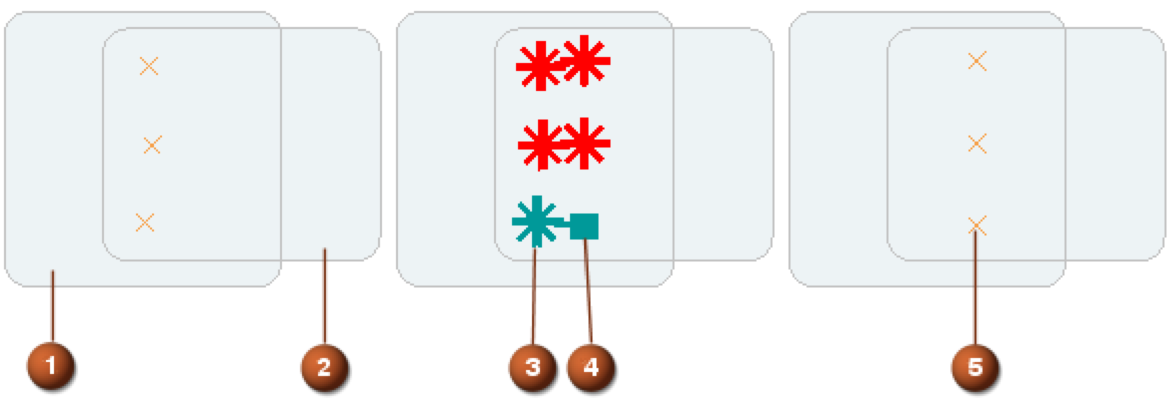
Move spot welds using Easy Spot

What is it?

You can now move existing spot weld features to new locations using the new **Move** option in the **Easy Spot** dialog box.

This option is useful if you have existing weld features and the underlying part geometry has changed. Use the option to compute new centerlines and map existing weld features to the new locations.

All the original attributes of the weld are maintained after it is moved.



1. First component being welded
2. Second component being welded
3. Original position of the weld points
4. New position of the weld points
5. Moved weld

When you select the weld points to be moved in the graphics window, the selected weld points appear in:

- A red color in the graphics window.
- A list in the **Easy Spot** dialog box.

  If you select a weld point in this list, it appears in the graphics window in a different color.

  Tip You can customize these colors using the **Dimension Color** and **Preview** Modeling customer defaults. These defaults are available in the **File® Utilities® Customer Defaults** dialog box, on the **Modeling® General® Colors** tab.

When you use the **Move** option, you can specify a **Reuse Match Tolerance** value for the new location.

The weld points you select for moving are listed in the **Easy Spot** dialog box as one of the following:

- **Mapped Point** — This indicates that the weld point is outside the specified **Reuse Match Tolerance** value and will be moved.
- **Coincident Point** — This indicates that the weld point is within the specified **Reuse Match Tolerance** value and will not be moved.
- **New Point** — This indicates that no mapping feature is assigned, and a new weld feature will be created.

You can also specify a **Uniform Spacing Tolerance** value. Specify this value as **0.0** to create a spacing less than the **Maximum Spacing**. This tolerance creates one less point on each overlap region that is computed.

Why should I use it?

Because this command computes the new centerlines and maps the existing weld features to the new locations, the process of locating the new positions for existing weld points is simplified.

Where do I find it?

Application	Modeling
Toolbar	**Weld Assistant® Easy Spot**
Menu	**Insert® Welding® Easy Spot**
Location in dialog box	**Type** group® **Move**

Import/Export Template Separator String

What is it?

You can now substitute attribute titles in weld point CSV files using separator strings in templates. Use the **Import/Export Template Separator String** customer default to define the separator string.

You can also specify the template file when you import a weld points file.

Example Consider a .def file with the following contents:

NUMBER OF SHEETS WELDED
X_POS
Y_POS
Z_POS

The exported template file looks like the following example.

NUMBER OF SHEETS WELDED	X_POS	Y_POS	Z_POS
2	43.5	21.956708	0
2	43.5	57.276736	0
2	43.5	95.599374	0

If you use a colon as a separator string and customize the template as follows:

NUMBER OF SHEETS WELDED
X_POS:x coordinate
Y_POS:y coordinate
Z_POS:z coordinate

The exported template file looks like the following example.

NUMBER OF SHEETS WELDED	x coordinate	y coordinate	z coordinate
2	43.5	21.956708	0
2	43.5	57.276736	0
2	43.5	95.599374	0

Tip Use a string of two or more characters to make your separator string unique. For example, **:::** or **:->**. This is useful when the attributes have colons such as **PART1:A**.

Where do I find it?

Application	Modeling
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Weld Assistant**® **Extras** page® **General** tab

Rules-based Structure Welding

Structure Welding

What is it?

Structure Welding is a new suite of welding functions that are designed to generate arc welds between prismatic type parts such as plates and extruded profiles.

Use rules-based structure welding to:

- Define arc welds in a CAD model.
- Generate a light weight NX feature to represent each weld.
- Control weld placement using algorithms that analyze the geometry being welded and identify the most logical location.
- Define welds by using commands that are available on the **Structure Welding** toolbar or by using rules-based callbacks. You can customize these callbacks to use any set of weld rules to control the weld definition.
- Drive the edge preparation and weld symbol creation from the welding joint feature.

Structure welding functions include the following:

-  **Welding Joint** — creation and modification of welding joints.

-  **Assign Weld Attributes** — edit and assignment of welding attributes to curves or edges.

-  **Weld Preparation** — preparation of edges of plates for welding.

- **Auto Weld Symbol** — creation of welding symbols.

- Rules-based callbacks.

Why should I use it?

Use these functions for prismatic type parts such as plates and extruded profiles.

Each weld feature is saved as a light weight NX feature. This allows you to define and work with a large number of welds in an NX session.

You can customize the callbacks to use a set of weld rules that comply with the standards used in your company.

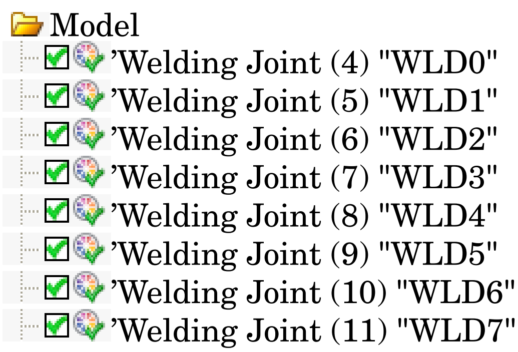
Where do I find it?

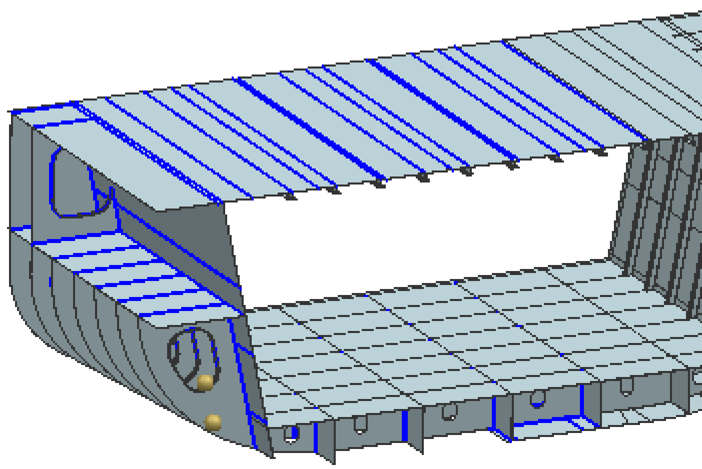
The callback location is *ugweld\samples*.

Application	Modeling
Toolbar	**Structure Welding**® [Structure welding command]
Menu	**Insert**® **Structure Welding**® [Structure welding command]

Welding Joint

Use the **Welding Joint** command to create T-joint and butt welds between plate-like structures. You can use this command to create welding joint features using lightweight connections when a solid representation of the weld is not required.





You can create welding joint features:

- Automatically. NX infers the weld edges based on the edge selections you make.
- Manually. You must select all the faces and edges required to define the welding joint feature.
- Using attributes placed on plates in the Ship Design application.

You can also:

- Display welding joints as curves. The curves are available for selection.
- Save the welding joint curves in Teamcenter.
- Configure the attributes assigned to the welding joints.
- Customize the color of the curves that represent these welding joint features by using the **Fillet Color** and **Butt Color** customer defaults.
- Create individual weld joints or create joints that are component pairs.

- Use the weld joints to prepare edges by using the **Weld Preparation** command.
- Create weld symbols for the weld joints.
- Reduce user-interaction when defining weld joints.
- Create multiple welds simultaneously.

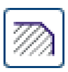
Where do I find it?

Welding Joint command

Application	Modeling
Toolbar	**Structure Welding® Welding Joint** 
Menu	**Insert® Structure Welding® Welding Joint**

Fillet Color/Butt Color customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Structure Welding® Common® Welding Joint** tab



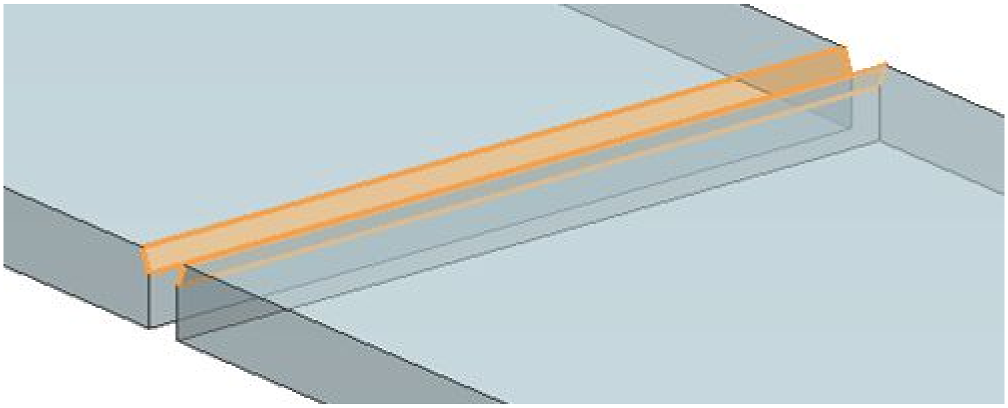
Weld Preparation

What is it?

Use the **Weld Preparation** command to prepare the edges of plates before welding.

This command must be used in conjunction with welding joint features. Welding joint features contain the information necessary to modify the edges of components being joined. The **Weld Preparation** command applies the modifications to the edges that are to be welded. The modifications are applied to the manufacturing body.

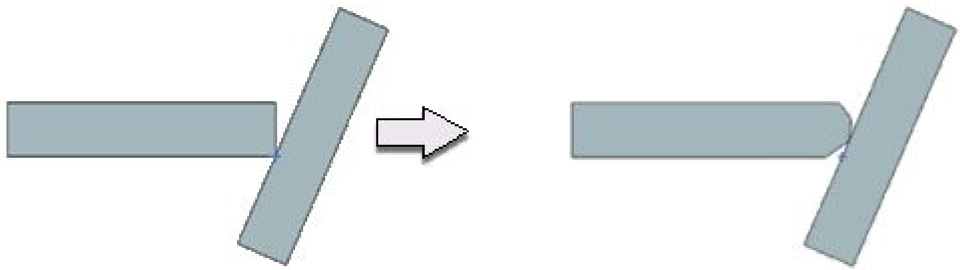
For example, you can chamfer the edges on one or both sides of a plate to prepare it for welding.



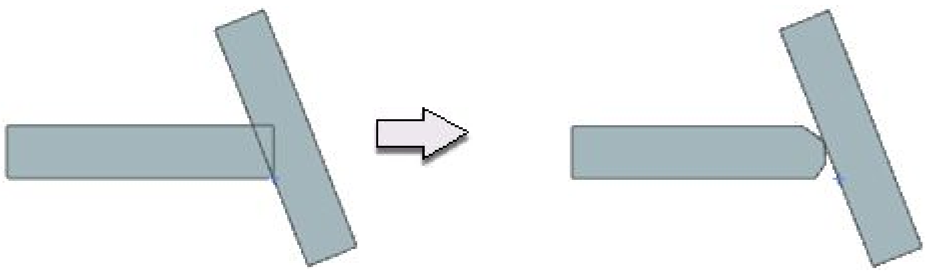
You can modify the profile of an edge or plate according to custom parameters. APIs are provided to specify the custom parameters.

The following are examples of the kinds of changes you can make.

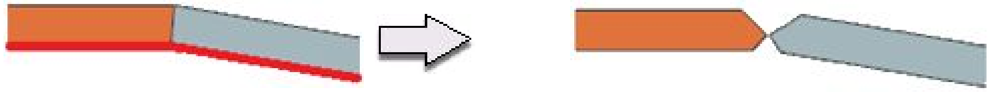
Example 1



Example 2



Example 3



Why should I use it?

The resultant features from this command can be used to convey weld preparation information on the model or on drawings.

Where do I find it?

Application	Modeling
Toolbar	**Weld Assistant® Weld Preparation**
Menu	**Insert® Welding® Weld Preparation**

Assign Weld Attributes

What is it?

Use the **Assign Weld Attributes** command to assign or edit welding attributes to curves or edges.

When you use this command on curves or edge, weld attributes are propagated as follows.

- If such curves are used to create plates in the Ship Design application, then the weld attributes propagate to the plate edges.
- If such edges are used to create a welding joint, then the weld attributes propagate to the welding joint.

Note You can restrict the attributes that are available for assignment using customer defaults.

Why should I use it?

Use this command to update edges or curves so that they can be used to define weld locations. The attributes you assign to the edges are transferred to the welding joint itself.

Where do I find it?

Assign Weld Attributes command

Application	Modeling
Toolbar	**Structure Welding® Assign Weld Attributes**
Menu	**Insert® Structure Welding® Assign Weld Attributes**

Characteristics customer default

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Structure Welding® Common® Welding Joint** tab

Callbacks for creating welding joint definition

What is it?

NX now provides APIs to write callbacks that communicate with a customer's rule-base for creation of welding joints.

You can:

- Use these APIs to define welding joints as per custom rules using NX.
- Use any of the NX Open Automation Languages such as Open for C/C++, Open for Java, Open for .NET, Knowledge Fusion, and Open C to write callbacks.
- Use the **Welding Joint** command to execute a defined callback and store the welding joint parameters with the welding joint feature. The **Weld Preparation** command uses these values to prepare the edges on the manufacturing body.

Where do I find it?

The callback location is *ugweld\samples*.

Chapter

10 Data translation

CATIA V5 Export/Import — CATIA V5 R20 SP3 support

What is it?

The NX CATIA V5 translator now supports the translation of CATIA V5 Revision 20 SP3 files to NX.

Why should I use it?

Use this option if you want to translate files created with CATIA V5 R20 SP3.

Where do I find it?

Support for CATIA V5 R20 SP3 files is available from all NX 8.0 CATIA V5 Translator interfaces.

The translator can be invoked from:

- command line
- the XlatorUI external user interface
- Teamcenter UI

or

Application	NX user interfaces.
Menu	**File® Open® Files of type:** CATIA V5 Files (*.CATPart) or CATIA V5 Assemblies (*.CATProduct). or **File® Import® Part® Files of type:** CATIA V5 Files (*.CATPart) or CATIA V5 Assemblies (*.CATProduct). or **File® Import® CATIA V5** or **File® Export® CATIA V5** or **File® Save as type:** CATIA V5 Files (*.CATPart) or CATIA V5 Assemblies (*.CATProduct).

DXF/DWG import interface

What is it?

The NX DXF/DWG translator import interface now has the following capabilities.

1. Ability to **Preview** (with zoom and different backgrounds) the selected DWF/DWG files.

2. Ability to select the template part file.

3. Options to select the data for import (model / layouts).

4. Options to control destination of selected data (modeling / drawing sheet / drawing view).

5. Options to create sketch curves (for future edits).

6. Option to create customer symbol of AutoCAD blocks.

7. Options to control layer mapping.

8. User interface to define text fonts, line style and cross hatch mapping.

Why should I use it?

Use this option if you want to configure the import of DWF/DWG data to NX.

Where do I find it?

Application	NX
Menu	**File® Import® DXF/DWG** **File® Open® DWG® Options**

PMI support in STEP translator

What is it?

You can now translate PMI data from NX to STEP as polyline (presentation) data. You can also now import PMI (polyline form) from STEP to NX.

Why should I use it?

Use this if you want to translate NX PMI to STEP and **Import** STEP PMI to NX.

Where do I find it?

Application	NX
Menu	**File® Import® STEP203** **File® Import® STEP214** **File® Export® STEP203** **File® Export® STEP214**

DXF/DWG Translator: Export to Drawing CGM data to DXF/DWG

What is it?

A new settings file keyword **EXPORT_DRAWING_USING_CGM** has been added to export NX Drawing to DXF/DWG via CGM.

This functionality will export NX model data to ACAD model space and CGM of selected drawing sheets to ACAD layouts.

Why should I use it?

Use this option is you want to create DXF/DWG based on the display (mostly for viewing/printing). It will allow you to create DXF or DXG based on the CGM representation of the NX drawing.

Where do I find it?

The new keyword **EXPORT_DRAWING_USING_CGM** is present in the settings file *dxfdwg.def* within *$DWFDWG_DIR*. The default value is set to No. Set the value to Yes if you want to take advantage of this functionality.

DXF/DWG Translator: Retaining the original aspect ratio of text

What is it?

A new settings file keyword **USE_NX_TEXT_ASPECT_RATIO** has been added to retain the same aspect ratio as that of NX Text in DXF/DWG.

Why should I use it?

While translating text from NX to DXF/DWG, because of the different fonts in NX and AutoCAD, the DXF/DWG translator does an automatic calculation of aspect ratio in order to keep the length of text the same as in AutoCAD. If you want to retain the same aspect ratio as in NX you can use the new settings file keyword **USE_NX_TEXT_ASPECT_RATIO**.

Where do I find it?

The new keyword **USE_NX_TEXT_ASPECT_RATIO** is present in the settings file *dxfdwg.def* within *$DWFDWG_DIR*. The default value is set to No. Set the value to Yes if you want to take advantage of this functionality.

DXF/DWG Translator: Translation of unsaved data

What is it?

You can now translate the unsaved data in your NX session to DXF/DWG using **File® Export® DXF/DWG**.

Why should I use it?

Use this option if you want to translate unsaved data to DXF/DWG.

Where do I find it?

If you export unsaved data to DXF/DWG, you will receive a warning asking to save the part first. If you click **Continue** it will translate the unsaved data to DXF/DWG.

Note Make use of the **Continue** option only when it is absolutely necessary. The graphics updates on screen may cause an increase in the translation time. We recommend working on saved drawings.

2D Exchange Translator: Translating multiple drawings

What is it?

You can translate multiple drawings using 2D Exchange to NX Part files in DXF, DWG, and IGES formats. This is possible only while exporting to Drafting.

Why should I use it?

Use this option if you want to translate multiple drawings to NX Part files in DXF, DWG, and IGES formats.

Where do I find it?

Application	NX user interface
Menu	**File® Export® 2D Exchange**, **Data to Export** tab, you can select multiple drawings under **Drawings Export**.

Note Exporting multiple drawings is not possible when you export data to Modeling.

NX CATIA V5 Translator: MAC operating system (OS) support

What is it?

The NX CATIA V5 translator is now available on MAC operating system (OS).

Why should I use it?

Use this option if you want to translate data between NX and CATIA V5 on the MAC OS platform.

Where do I find it?

The translator can be invoked on MAC OS from:

- command line
- the XlatorUI external user interface
- Teamcenter UI

or

Application	NX user interface
Menu	**File**® **Open**, **File**® **Import**® **Part**, under **Files of type:** CATIA V5 Files (*.CATPart) or CATIA V5 Assemblies (*.CATProduct) or **File**® **Import**® **CATIA V5** or **File**® **Export**® **CATIA V5**

Chapter

11 *Programming and automation*

Multiple libraries for linking in C++ projects

What is it?

When you perform linking in C++, you now have to include additional libraries. Due to the continuous addition of JA functions, the **libnxopencpp** library is reaching its limit, so the **libnxopencpp** library is split into multiple libraries. The libraries are based on the namespaces available. There are currently 35 different namespaces in NX Open, so 35 new libraries are added.

You need to refer to the appropriate libraries when building a new NX Open C++ application, or when you modify existing projects to refer to the newly added libraries.

Note Only linking is affected. The compilation process for NX Open programs remains the same.

Why should I use it?

You need to incorporate the new library files into your C++ projects.

Where do I find it?

The NX Open C++ wizard and *uflink* script are updated for the new libraries.

Switching to another application within an NX Open program

What is it?

The **ApplicationSwitchRequest** API is added to enable you to switch to another application within an NX Open program. This allows you to build customizations that access multiple NX applications.

Due to existing coding limitations, the API is used to request the application switch within the program but the actual switch does not occur until the NX Open program terminates.

As long as the current file is a standard **.prt** file and the user has the appropriate license, the application switch can be performed. If the current file is a **.sim** or **.fem** file and the current application is Simulation, then the application switch cannot be performed.

Why should I use it?

You can switch to another application in your NX Open programs.

Where do I find it?

The new API is described in the NX Open API reference documentation.

Java objects supported when using remote applications

What is it?

UF wrapped Java methods that use UFVariant objects as parameters or return types are now supported when they are used in applications that are accessed remotely.

Why should I use it?

You can use the UFVarient objects in applications that are used remotely.

Where do I find it?

The JAVA APIs that use UFVariant objects are described in the NX Open API reference documentation.

Block UI Styler

Balloon tooltip properties for dialog blocks

What is it?

You can use balloon tooltip properties to provide the end-user with additional information for dialog blocks that you create using NX Open and Block UI Styler. The tooltip appears when the user points to an option or icon in the block. The balloon tooltip properties allow you to specify the text, the image, and the layout of text with respect to the image in the balloon tooltip.

These properties are available for many dialog blocks. Blocks support properties for single or multiple balloon tooltips.

Note The user must select the **Show Balloon Tooltips on Dialog Options** check box in the **Customize** dialog box to turn on the balloon tooltips. There is no special property to turn on balloon tooltips; the tooltip appears when you specify text and an image for it.

Blocks with a single balloon tooltip

The following blocks support a single balloon tooltip:

- Label/Bitmap
- Action Button
- Object Color Picker
- RGB Color Picker
- Specify Point
- Specify Vector
- Section Builder
- Specify CSYS
- Specify Plane
- Specify Orientation
- Select Object
- Select Feature
- Curve Collector
- Body Collector
- Face Collector
- Super Section
- Double
- Integer
- Expression
- Angular Dimension
- Radius Dimension
- Linear Dimension
- On Path Dimension

The balloon tooltip appears over the label or the bitmap for the block.

Property Name	Description	Access	Property Type	List of Values
BalloonTooltipText	Specifies text for the balloon tooltip.	CIGS	String	Any string value. Type \n to specify a line feed in the text.
BalloonTooltipImage	Specifies the file name of the image to be displayed in the balloon tooltip.	CIGS	String	Specify a valid bitmap file name from the NX library. You need not specify the suffix and extension. You can also specify the complete path of an image file. NX supports the following image file formats: TIFF, PNG, GIF, JPEG, and BMP.
BalloonTooltipLayout	Specifies the location of text with respect to the image in the balloon tooltip. This is applicable only when both text and image are specified.	CIGS	Enum	Horizontal - This is the default value. The text is placed to the right of the image. Vertical - The text is placed below the image. Specify this value when the image is more than 300 pixels wide or if the width to height ratio of the image is greater than 4.

Blocks with two balloon tooltips

The Toggle block supports separate balloon tooltip properties for on and off states. Therefore, different text and images can appear in the balloon tooltip for on and off states. The balloon tooltip appears over the label, button, or bitmap for the toggle block. The following table describes the balloon tooltip properties that are available for the Toggle block.

Property Name	Description	Access	Property Type	List of Values
BalloonTooltipOnText BalloonTooltipOffText	Specifies text for the balloon tooltip.	CIGS	String	Any string value. Type \n to specify a line feed in the text.
BalloonTooltipOnImage BalloonTooltipOffImage	Specifies the file name of the image to be displayed in the balloon tooltip.	CIGS	String	Specify a valid bitmap file name from the NX library. You need not specify the suffix and extension. You can also specify the complete path of an image file. NX supports the following image file formats: TIFF, PNG, GIF, JPEG, and BMP.
BalloonTooltipLayout	Specifies the location of text with respect to the image in the balloon tooltip. This is applicable only when both text and image are specified.	CIGS	Enum	Horizontal - This is the default value. The text is placed to the right of the image. Vertical - The text is placed below the image. Specify this value when the image is more than 300 pixels wide or if the width to height ratio of the image is greater than 4.

The Specify Axis block supports separate balloon tooltip properties for the two sub-blocks: Specify Vector and Specify Point. The balloon tooltip appears over the label of the Specify Axis sub-blocks. The following table describes the balloon tooltip properties available for the Specify Axis block.

Property Name	Description	Access	Property Type	List of Values
BalloonTooltipVectorText BalloonTooltipPointText	Specifies text for the balloon tooltip.	CIGS	String	Any string value. Type \n to specify a line feed in the text.

BalloonTooltipVectorImage BalloonTooltipPointImage	Specifies the file name of the image to be displayed in the balloon tooltip.	CIGS	String	Specify a valid bitmap file name from the NX library. You need not specify the suffix and extension. You can also specify the complete path of an image file. NX supports the following image file formats: TIFF, PNG, GIF, JPEG, and BMP.
BalloonTooltipLayout	Specifies the location of text with respect to the image in the balloon tooltip. This is applicable only when both text and image are specified.	CIGS	Enum	Horizontal - This is the default value. The text is placed to the right of the image. Vertical - The text is placed below the image. Specify this value when the image is more than 300 pixels wide or if the width to height ratio of the image is greater than 4.

Blocks with a list of balloon tooltips

The Enumeration block and String block with Combo Box presentation provide a list of values. Therefore, these blocks have a list of balloon tooltip properties, so that a value can be specified for each item in the list. The balloon tooltip for each item appears when you point to the item in the list.

Property Name	Description	Access	Property Type	List of Values
BalloonTooltipTexts	Specifies balloon tooltip text for each item in the list. Each item in the list must have a corresponding text value.	CIGS	String	List of any string values. Type NULL for items that do not have any text value. Type \n to specify a line feed in the text.

BalloonTooltipImages	Specifies the file name of the image to be displayed in the balloon tooltip corresponding to an item in the list. Each item in the list must have a corresponding value.	CIGS	String	Specify a valid bitmap file name from the NX library. You need not specify the suffix and extension. You can also specify the complete path of an image file. NX supports the following image file formats: TIFF, PNG, GIF, JPEG, and BMP. All images must have the same size. Type NULL for items that do not have a corresponding image.
BalloonTooltipLayout	Specifies the location of text with respect to the image in the balloon tooltip. This is applicable only when both text and image are specified.	CIGS	Enum	Horizontal - This is the default value. The text is placed to the right of the image. Vertical - The text is placed below the image. Specify this value when the image is more than 300 pixels wide or if the image width to height ratio is greater than 4.

Enable OK/Apply Button

Code Generation options

Enable OK/Apply Button	This callback is invoked by the dialog box that requests whether to enable the **OK** and **Apply** buttons.

Callbacks for Automation code

Enable OK/Apply Button	This callback allows NX to enable or disable the **OK** and **Apply** buttons. If the dialog box contains a **Selection Block** and no selections are specified, the dialog box will disable the **OK** and **Apply** buttons even if the callback returns the value as **True**. Callback return type — Boolean. Possible return values —**TRUE** enables the **OK** and **Apply** buttons, and **FALSE** disables the **OK** and **Apply** buttons.

User Defined UI Block

What is it?

Block UI Styler now supports User Defined UI Block (UDB). You can:

- Create a UDB using the blocks available on the **Block Catalog** palette. The automation code and catalog file are generated and newly created UDB is automatically added to the palette.
- Reuse UDB that is previously saved in the pallete and insert it in dialog boxes like any other block.
- Create a UDB that contains another UDB. NX allows multiple levels of reusability.

A block defines the UI constituents. The automation code defines the behaviour of the UDB and the catalog file defines the placement of the block on the palette. The automation code for the dialog box is built with the automation code used for the UDB automation code (or library) to create an application library. The automation code for the dialog box and the UDB must be in the same language.

The file extensions are as follows:

- Block — .dlx
- Automation code — .vb, .cs, .cpp, or .java based on the code generation option
- Catalog file — .udx

The application library, the library of blocks, and the dlx files for the dialog box and UDB complete the application. The dialog box is launched when the application library is executed.

NX allows multiple catalog files for better catalog management. The location of these catalog files follows the same rules as specified for the dlx file for the dialog box.

Where do I find it?

Application	Block UI Styler
Menu	**Start**® **All Applications**® **Block UI Styler**
Location in dialog box	**Dialog** dialog box® **Type** group® from the list, select **User Defined UI Block**

Knowledge Fusion

Automation Package Deployment

What is it?

This enhancement lets you use Teamcenter to deploy or upgrade automation applications such as Knowledge Fusion, Check-Mate, or NX Open in NX. An automated deployment or update must have the following components:

- The compressed automation package files saved in Teamcenter as an item revision dataset.
- A list of item numbers for the automation packages saved in a text file.
- An environment variable that will be turned on by the deployment administrator every time a deployment or an update needs to occur.

After these components are in place and the environment variable is turned on, the deployment starts when users launch NX.

Cloning Knowledge Fusion components

What is it?

This enhancement automatically updates the values of the **File_Name** attribute in appropriate Knowledge Fusion instances when you clone, rename, import, or export an assembly. The updated attribute values refers to the file name of the newly created part files.

Where do I find it?

Application	Assemblies
Prerequisite	You must have Knowledge Fusion components in the assembly.
Menu	**Assemblies**® **Clone**® **Create Clone Assembly** **File**® **Import** **File**® **Export** **File**® **Save As**

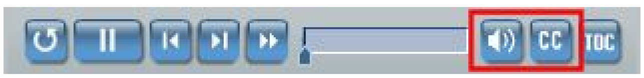
Chapter

12 Videos with audio

Videos with audio

Beginning with this release, the video examples included with the Help now include voice-over narration. This narration provides important information about the steps being demonstrated.

For each video, you can use the options on the play bar to display closed caption text or to mute the narration.



Sample videos:

Where do I find it?

Videos are available in help topics where you see the  **Video** button or from the **Video examples** page in the Design, Manufacturing, and Advanced Simulation Help.