What's New in NX 8.5 Beta

Properietary and Restriced Rights Notice

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Chapter

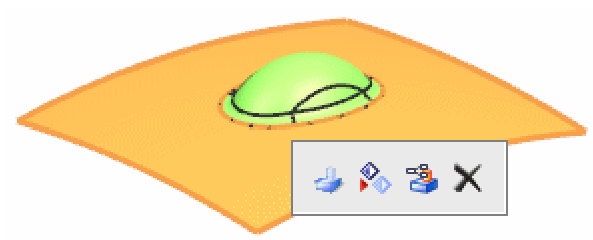
1 Fundamentals

Context toolbar

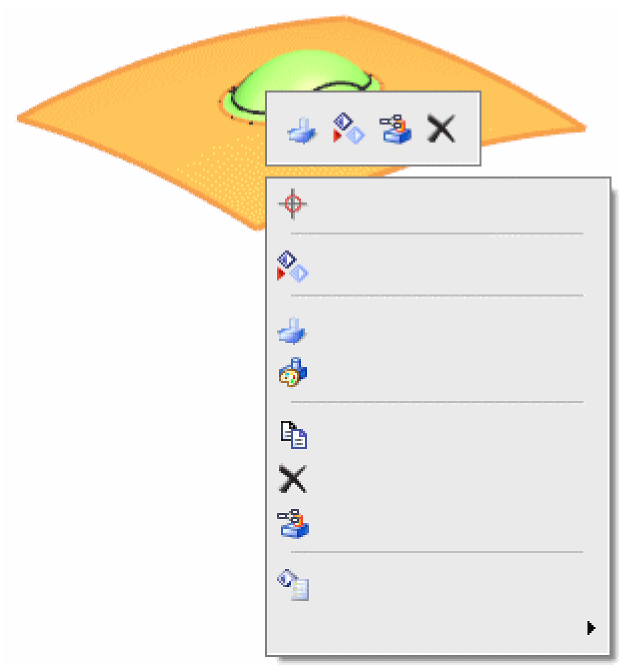
What is it?

A context toolbar is a toolbar that contains the commands that you are most likely to use for the object or objects that you select. If you select one or more objects of the same type, the context toolbar displays the commands that are specific to the selected objects. If you select multiple objects of different types, NX displays a generic context toolbar that can contain commands that you can use on all the selected object types.

NX displays only the context toolbar when you click one or more objects in the graphics window.



NX displays the context toolbar with the context menu when you right-click one or more objects in the graphics window, the **Part Navigator** or the **Assembly Navigator**.



You can also, customize the object-specific and generic context toolbars.

Display view commands in the context menu

What is it?

You can customize the context menu by adding or removing the view operation commands to it.

If you select the **Show View options on all context menus** check box, the context menu displays the **View** cascading menu with generally used view operation commands. By default, this check box is not selected.

Note This change applies to all the context menus.

Where do I find it?

Toolbar	Click on the **Toolbar Options** (down arrow) at the end of the toolbar, and then choose **Add or Remove Buttons® Customize**
Menu	**Tools® Customize® Customize Toolbars** tab **File® Utilities® Customer Defaults**
Location in the **Customer Defaults** dialog box	**General® User Interface® General** tab

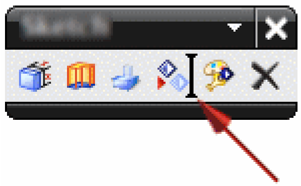
Customizing the context and radial toolbars

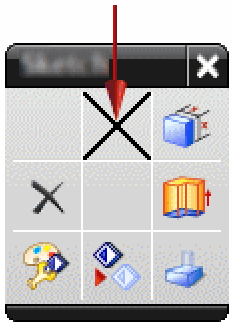
What is it?

You also customize the context and radial toolbar as per applications or tasks for single or multiple object type selection.

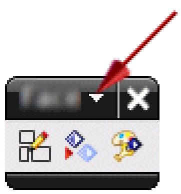
To customize context and radial toolbar for single object type selection:

- Drag the command from the toolbar, menu, or **Commands** tab of the **Customize** dialog box and drop it on the respective toolbar you want to customize.





- Use the **Add or Remove Buttons** from the down-arrow of the context toolbar title area.



Note To customize a context or radial toolbar, you need to select an object in the graphics window while the **Customize** dialog box is open.

To customize context toolbar for multiple object type selection:

Select the **All Objects** entry in the **Selected Objects** list of the **Customize** dialog box and add the required commands to the displayed generic context toolbar.

> **Note** The commands added to the **All Objects** entry are appended to all the context toolbars irrespective of the selected object type.

The customization of the context and radial toolbar is stored only in the current application. You must enter specific application to customize the context and radial toolbar for that application.

Where do I find it?

Toolbar	Click on the **Toolbar Options** (down arrow) at the end of the toolbar, and then choose **Add or Remove Buttons® Customize**
Menu	**Tools® Customize**
Location in dialog box	**Customize Toolbars** tab

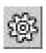
Linux — standard buttons relocated

What is it?

The following buttons are relocated:

 **Pin**

 **Reset**

 **Show Menu/ Hide Menu**

 **Help**

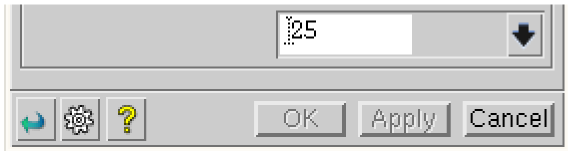
Resource bar and **QuickPick** dialog box

- The **Pin**  button is now located at the top of the Resource bar.

- The **Pin**  button is removed from the **QuickPick** dialog box. You can fix the location of the **QuickPick** dialog box using the new **Lock Dialog Position** selection preference.

Dialog boxes

The **Reset**, **Show Menu/ Hide Menu**, and **Help** buttons are now located at the lower left-hand corner at the bottom of the dialog box.



Note Buttons appear in a dialog box only when their function is supported by the command.

Importing points

What is it?

Use the **Points** command to import point data stored in ASCII files.

Why should I use it?

Use this command to import a point data file for reverse engineering purposes.

Where do I find it?

Menu	**File® Import® Points**

Read attributes from JT files into NX

What is it?

While reading JT files, NX attaches JT node attributes to either the NX part or the respective bodies within the part, depending on what the JT node represents. In addition, it names the NX objects and any corresponding features based on the names of JT file nodes. This enhancement makes the relationship between the data in NX and the original JT data more obvious.

User defined object names in NX

What is it?

The length of object names in NX is now increased to 128 bytes.

Why should I use it?

You can use descriptive object names for easier identification during different stages of design.

Selection Preferences enhancement

What is it?

You can now use the **Lock Dialog Position** selection preference to fix the location of the **QuickPick** dialog box in the graphics window.

If you select this preference and then move the **QuickPick** dialog box to a new location, that location is fixed. When you reopen the **QuickPick** dialog box, it appears at the fixed location and not the default location near the selection point.

Where do I find it?

Application	Gateway
Menu	**Preferences® Selection**
Location in dialog box	**Selection Preferences® QuickPick** group

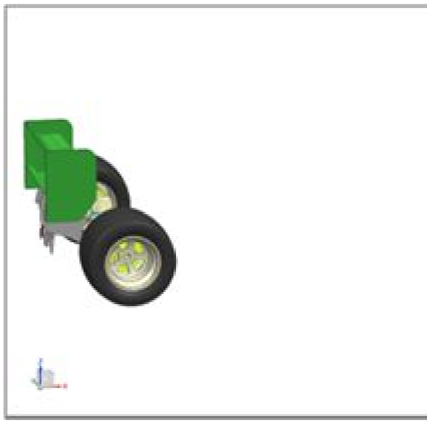
Fit enhancements

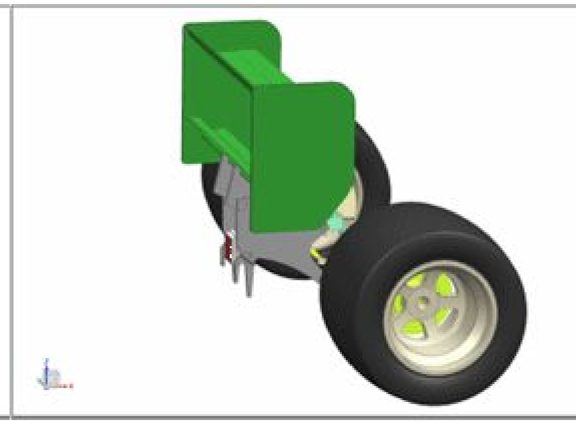
What is it?

You can now set the following Visualization preferences for the **Fit** command:

Fit on Show or Hide	Fits your model to the view automatically after you use the **Hide** or **Show** command.

Exclude Datums From Fit Fits your parts to the view without taking datum objects into consideration.

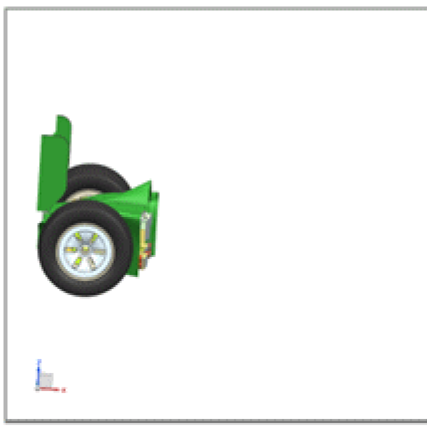




Exclude Datums From Fit turned off

Exclude Datums From Fit turned on

Fit to Work Section Fits the view to the clipped planes when view sectioning is displayed.



Fit to Work Section turned off

Fit to Work Section turned on

Where do I find it?

Toolbar	**Visualization® Visualization Preferences** 
Menu	**Preferences® Visualization**
Location in dialog box	**View/Screen** tab® **Session Settings** group® **Fit on Show or Hide** or **Exclude Datums From Fit** or **Fit to Work Section**

Set Rotation Reference

What is it?

Use the **Set Rotation Reference** command to select an alternative rotation point or rotation axis. This command replaces **Set Rotate Point** in the **View** shortcut menu.

Use the **Clear Rotation Reference** command to clear your previously selected rotation reference.

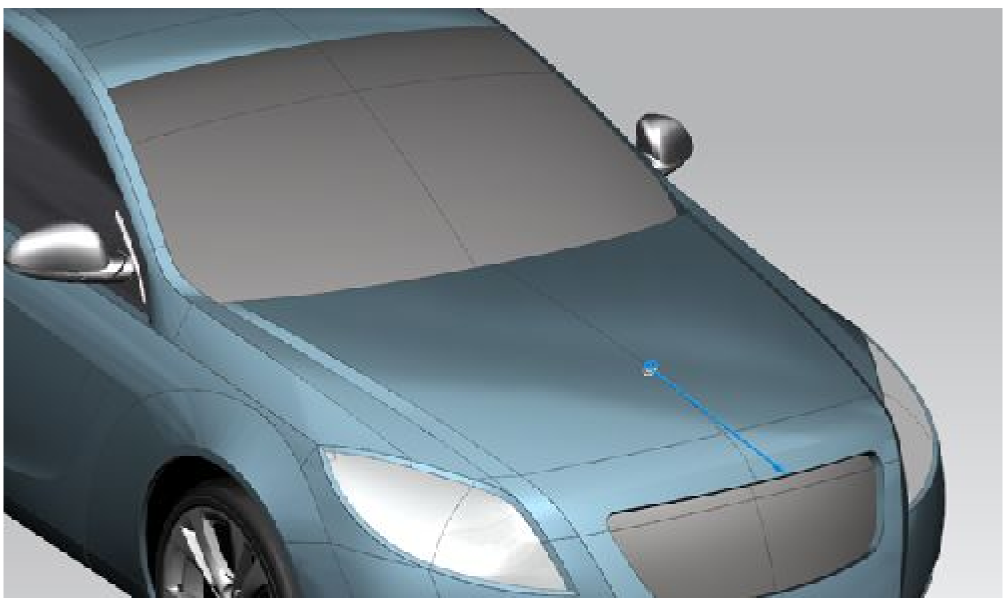
To set the axis of rotation, you can select an object such as:

- An edge or curve. The object can be either linear or non-linear.
- A datum axis.

Why should I use it?

You can rotate the display about a specific point or axis to view part features, assembly features and surfaces in the design.

Example In automotive design, you can use a vector as a rotation axis to evaluate surfaces along primary visual lines for a car's exterior body panel design.



Where do I find it?

Shortcut menu	Right-click in the background of the graphics window® **Set Rotation Reference**  or **Clear Rotation Reference**

Rotate enhancements

What is it?

During interactive view rotation in NX, the default center of view rotation and the display of the rotation center are now enhanced.

NX derives the default center of rotation from the visible geometry of the work part. If view sectioning is enabled, NX adjusts the rotation center, if necessary, to the unclipped region of the model.

The new **Display Center of Rotation** Visualization preference lets you display the center of rotation in the graphics window. The display color for the active center of rotation depends on your selection of the **Handles** Visualization preferences as follows:

- **Preselection** color is displayed when NX derives the center of rotation from visible geometry.

- **Active Handle** color is displayed when you select an alternative rotation point or axis using the **Set Rotation Reference** command.

- **Selection** color is displayed when you click and hold the middle mouse button to set a temporary center of rotation.

- **OrientXpress** color is displayed when the center of rotation is the center of the view.

To set the **Handles** Visualization preferences, choose **Preferences**® **Visualization**® **Handles** tab® **Part Settings** group.

These enhancements are applicable for interactive view rotations performed using the middle mouse button, or a 3D input device.

Example In the following graphic, the center of rotation is displayed in the **Active Handle** color, which is set to **Blue**.

Why should I use it?

These enhancements can significantly improve interactive display during rotation by keeping the model part visible rather than rotating it out of view.

Where do I find it?

Toolbar	**Visualization**® **Visualization Preferences**
Menu	**Preferences**® **Visualization**
Location in dialog box	**View/Screen** tab® **Session Settings** group® **Display Center of Rotation** check box

Align Facets along Edges

What is it?

The **Align Facets along Edges** option generates facets for solid and sheet bodies by aligning facets on both the sides of the common edge. This command aligns the facets along the common edges by sharing vertices for the generated facets.

You can display the generated aligned facets by selecting the **Show Facet Edges** check box.

While translating the NX files to JT, the aligned facets are also translated along with the material and texture applied to the facets.

Why should I use it?

Use this option to get a better quality rendering. This option results in longer rendering time to generate the facets.

Where do I find it?

Toolbar	**Visualization® Facet Settings** 
Menu	**View® Operations® Facet Settings** **View® Visualization® Faceting** tab **File® Utility® Customer Defaults® Gateway® Visualization® Faceting** tab

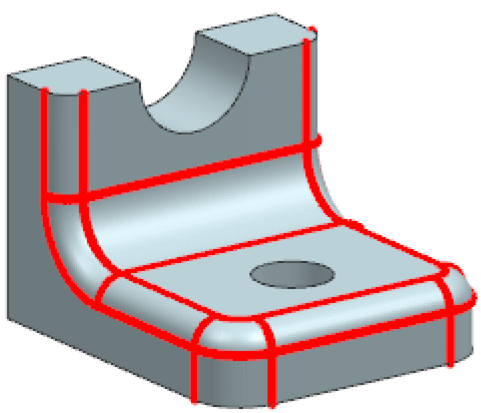
Smooth Edges visualization enhancement

What is it?

You can now use the following **Smooth Edges** Visualization preferences for the **Shaded with Edges** rendering style:

- **Color**
- **Font**
- **Width**

You can also control the **Angle Tolerance** value. NX determines whether an edge is smooth based on the **Angle Tolerance** value.



In the previous releases, you could use these preferences only for the **Static Wireframe** rendering style.

Why should I use it?

You can use these Visualization preferences during modeling tasks to quickly differentiate between smooth and sharp edges.

Example If you are blending a large collection of edges, you can differentiate immediately between the blended edges, and the edges that you still need to work on.

Where do I find it?

Toolbar	**Visualization**® **Visualization Preferences**
Menu	**Preferences**® **Visualization**
Location in dialog box	**Visual** tab® **Edge Display Settings** group® **Part Settings (Selected Views)** subgroup

Custom Widths enhancements

What is it?

You can now work with nine **Custom Widths** for objects in Printing, Plotting, PDF Export and CGM Import or Export. You can specify default **Custom Widths** using **Customer Defaults**.

In previous releases, you could work with three **Custom Widths**.

Where do I find it?

Menu	**File**® **Print**, or **Plot**, or **Export**, or **Import** **File**® **Utilities**® **Customer Defaults** **Preferences**® **Object** **Preferences**® **Visualization** **Edit**® **Object Display**

	Print dialog box® **Settings** group® **Width** **Plot** dialog box® **Color And Width** group® **Widths** list® **Custom Widths** **Export CGM** dialog box® **Properties** group ® **Widths** list® **Custom Widths** **Object Preferences** dialog box® **General** tab® **Width** list **Edit Object Display** dialog box® **General** tab® **Width** list **Customer Defaults** dialog box® **Gateway**® **Object**
Location in dialog box	**Visualization Preferences** dialog box® **Line** tab® **Show Widths** check box

See-Thru Preview

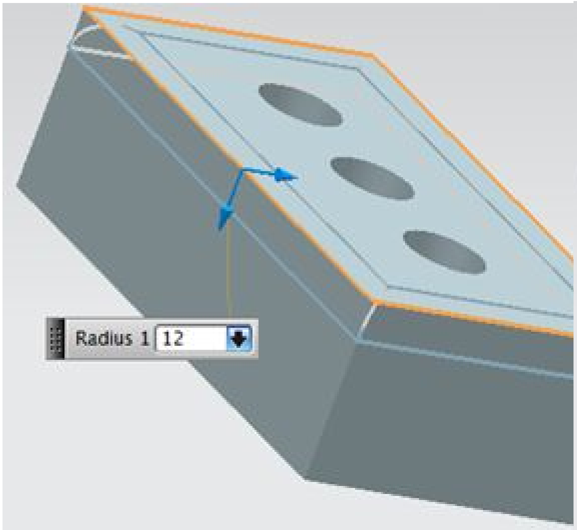
What is it?

When you create or edit a feature, NX automatically applies the **See-Thru Preview** to emphasize the feature in the model preview, and de-emphasize any other existing features and geometry. The emphasized features are also rendered as shaded display.

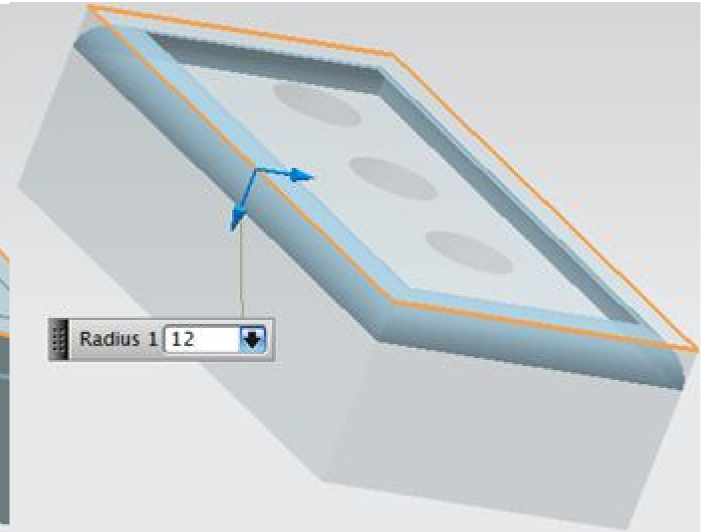
You can customize the de-emphasis of geometry less important to the command by changing one or more of the following Visualization preferences:

- Translucency
- Color
- Edge effects

Note Important geometry is the geometry which is being created or edited by the NX command.



See-Thru Preview turned off for Edge Blend



See-Thru Preview turned on for Edge Blend

Why should I use it?

Use **See-Thru Preview** to emphasize and better visualize the preview seen while creating features.

Where do I find it?

See-Thru Preview:

Toolbar	**View**® **See-Thru Preview**
Menu	**View**® **Visualization**® **See-Thru Preview**

Object emphasis Visualization preferences:

Toolbar	**Visualization**® **Visualization Preferences**
Menu	**Preferences**® **Visualization**
Location in dialog box	**Visualization Preferences** dialog box® **Emphasis** tab

True Studio 3D Dome environment

What is it?

A new **3D Dome** scene background is available when setting up scenes in the **True Studio** environment.

3D Dome uses a panoramic image that is mapped onto a hemisphere dome.

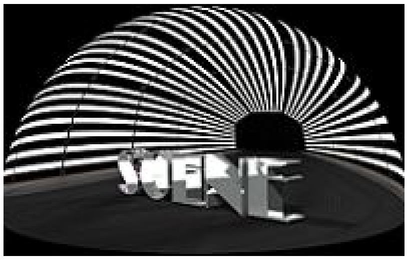
As you pan around the model, different views of the model combined with the dome background, reflect the three dimensional character of the environment, creating a more realistic scene.

At any desired scene rotation, High Quality Image static images can be created for the maximum realism of a model in a scene.

The **3D Dome** option can work in conjunction with the **Environmental Image** orientation settings on the **Global Illumination** page to define the dome environment.

There are four available 3D dome visualization scenes available on **System Scenes** palette.

Factory scene



Greenroom scene

Plaza scene – modern

Plaza scene – European, winter

Why should I use it?

Use a **3D Dome** background along with **Global Illumination** and a **Stage** shadow catcher floor plane for the most realistic real-time and static image renderings.

During real-time view rotation, your model will appear to be in a real-world environment.

Where do I find it?

Application	Gateway
Prerequisite	**Advanced Studio Display**
Toolbar	**Visualize Shape® Scene Editor**
Menu	**View® Visualization® Scene Editor**
Location in dialog box	**Background** tab® **Background** list, **3D Dome Panoramic LDR Image** group, **3D Dome Settings** group **Global Illumination** tab® **Environmental Image Orientation** group

Resource bar	**System Scenes** 

Measure Extremes

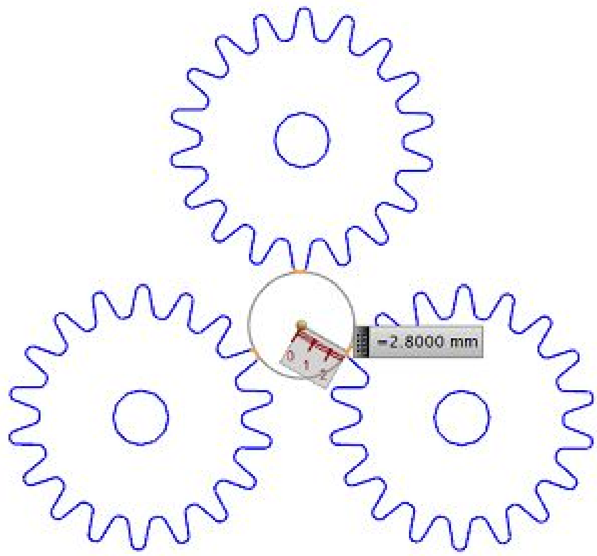
What is it?

Measure Extremes command calculates the farthest 3D point from the global origin relative to three directions, or maximum or minimum 2D radius and angle measurements on planar geometry.

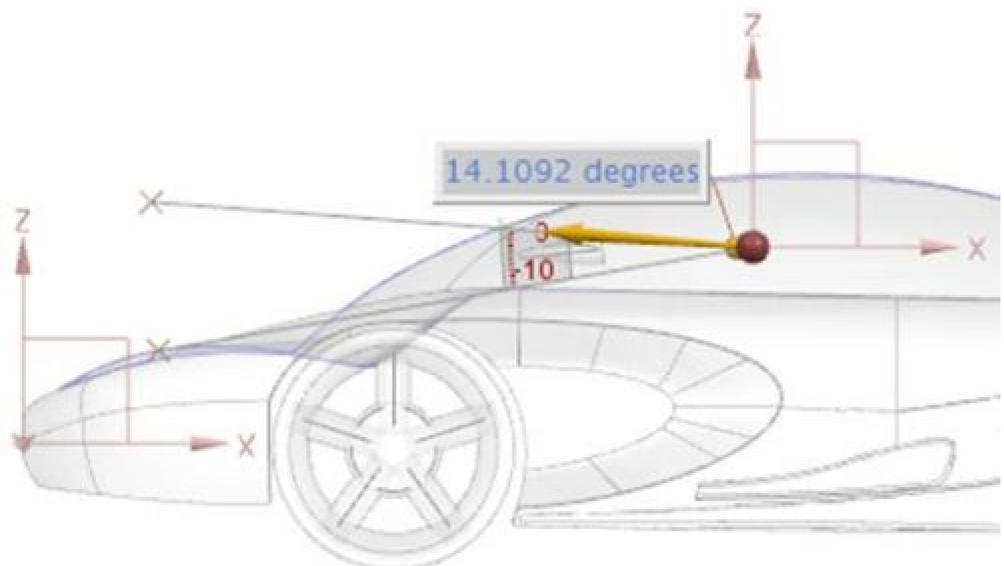
Using this command you can:

- Calculate the extreme point within a set of objects, by providing a maximum of three reference vectors.

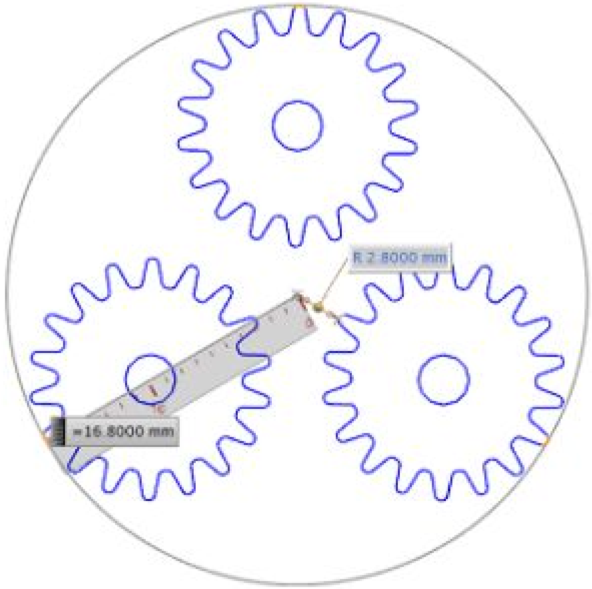
- Calculate the point at a minimum or maximum radial distance within a set of 2D coplanar objects, by providing a reference vector and a reference point as input.



- Calculate the point at a minimum or maximum angle within a set of 2D coplanar objects, by providing a reference vector and a reference point as input.



- Calculate the minimum or bounding radius to circumscribe a circle surrounding the 2D coplanar set of objects.



Why should I use it?

Use this command to identify objects that are located at certain extreme positions in a robust model.

Example You can use this command to identify the following:

- Combined overall size that must be packaged
- Maximum openings between objects
- Visible field of view angle between objects etc.

Where do I find it?

Application	Gateway
Menu	**Analysis® Measure Extremes**

Selection Intent with measurement commands

What is it?

The following commands are enhanced to store the selection intent rule with the measurement:

- **Measure Length**
- **Measure Face**

- **Measure Bodies**

Why should I use it?

With the enhancement in the **Measure Length**, **Measure Face**, and **Measure Bodies** commands to store the selection intent rule, the stored measurement now changes dynamically according to any changes in the input values if the original geometry that you measured changes.

Example Suppose you set the **Face Rule** on the selection bar to **Body Faces**, and use the **Measure Face** command. When you select a face, all the faces on the body are selected for measurement. The measurement is stored in the Part Navigator with the Body Faces rule. If you make a change which adds more faces to the body in the part history before the measurement feature, and then check the stored measurement, the measurement includes the new faces. Note that the measurement feature is at a particular timestamp order in the part history, and that changes made after the measurement feature will not be included in the measurement.

Where do I find it?

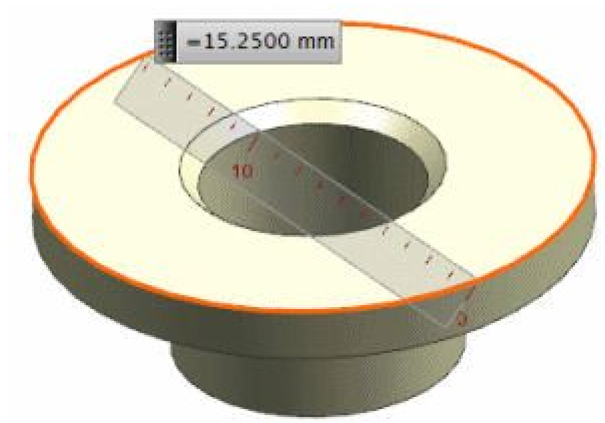
Application	Gateway
Toolbar	**Utility® Measure Length/ Measure Face/ Measure Bodies**
Menu	**Analysis® Measure Length/ Measure Face/ Measure Bodies**

Measure Distance enhancements

What is it?

The **Measure Distance** dialog box has three new Type options.

- **Diameter** measures the diameter of circular entities.



- **Between Objects Sets** measures the distance between two sets of objects that you select using the selection intent rule.

- **Projected Distance between Objects Sets** measures the projected distance between two sets of objects that you select using the selection intent rule. The distance is projected along a selected vector.

Where do I find it?

Application	Gateway
Toolbar	**Utility**® **Measure Distance**
Menu	**Analysis**® **Measure Distance**
Location in dialog box	**Type** group® **Measure Diameter/ Between Component Sets/ Projected Distance between Component Sets**

Simple Measure Favorites commands

What is it?

The **Simple Measure Favorites Drop-down** list provides the following commands that you can use to do non-associative measurements:

- **Simple Distance**
- **Simple Angle**
- **Simple Length**
- **Measure Radius**
- **Measure Diameter**

Why should I use it?

These commands have simplified dialog boxes that you can use to select the objects or points that you need to get a non-associative measurement.

Where do I find it?

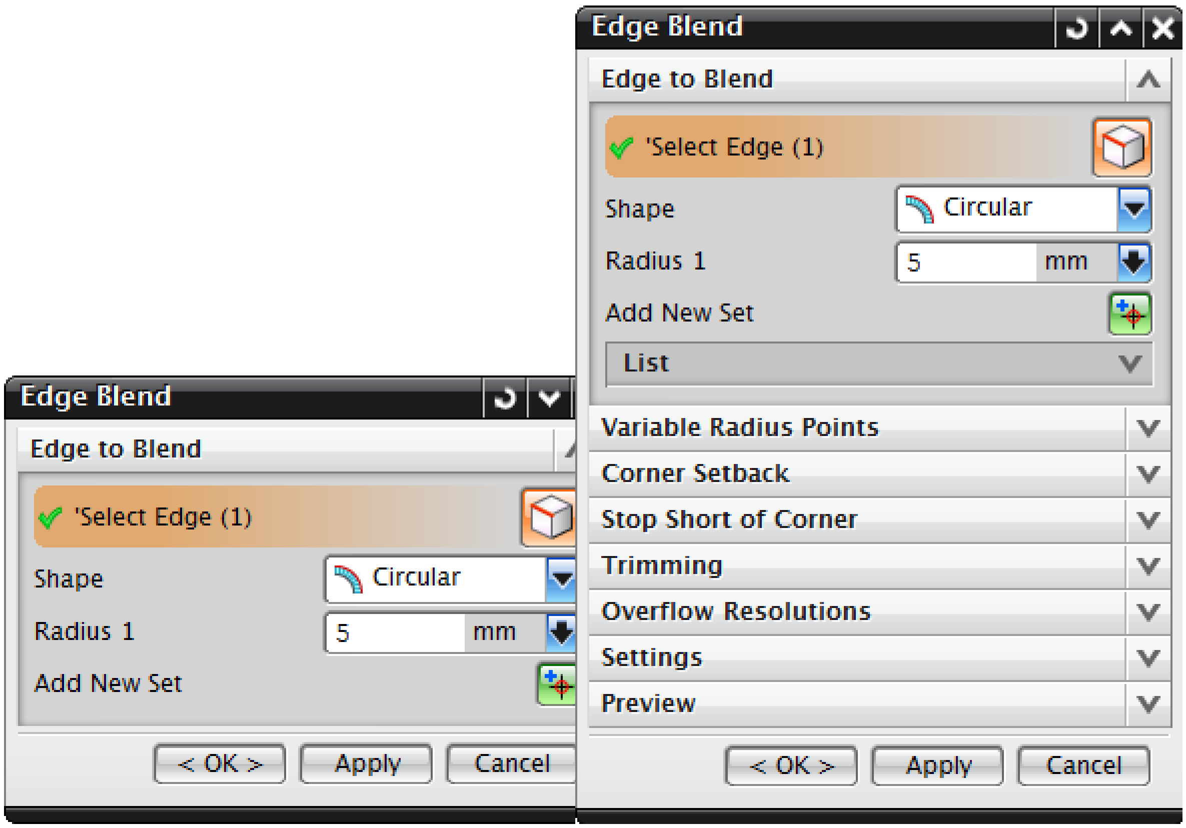
Application	Gateway
Toolbar	**Utility**® **Simple Measure Favorites Drop-down** list
Menu	**Analysis**® **Simple Measure Favorites Drop-down** list

Less and More dialog box versions

What is it?

NX displays Less and More versions of command dialog boxes that have at least one collapsed group in their full layout. The version that you see depends on your role.

The example shows Less and More versions of a command dialog box.



Click or on the dialog box title bar to switch between the Less and More versions of the dialog box.

You can control whether NX displays the Less or More version by default, using the **Command Dialog Version** User Interface Preference.

Tip To find the User Interface Preference, choose **Preferences**® **User Interface** and click the **General** tab.

You can create your own favorite versions and access the version you want from the context menu when you right-click the dialog box title bar.

You can also Press/Hold or on the dialog box title bar to get the full Favorites menu.

Why should I use it?

You can now have easy access to simplified or full versions of command dialog boxes. NX matches the complexity or simplicity of the command dialog box to your NX role.

Chapter

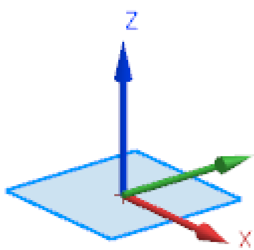
2 *CAD*

Sketching

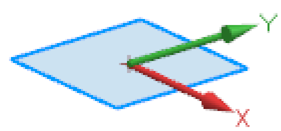
Sketch CSYS display enhancement

What is it?

In order to simplify the display while sketching, the internal datum CSYS no longer shows the sketch plane normal. The sketch internal datum CSYS is like any datum CSYS created, but when a sketch is active the sketch normal is no longer displayed.



NX8.0



NX8.5

Where do I find it?

Application	Modeling, Drafting, Shape Studio, Sheet Metal
Prerequisites	Sketch task environment, or while a sketch is active

Sketch CSYS naming enhancement

What is it?

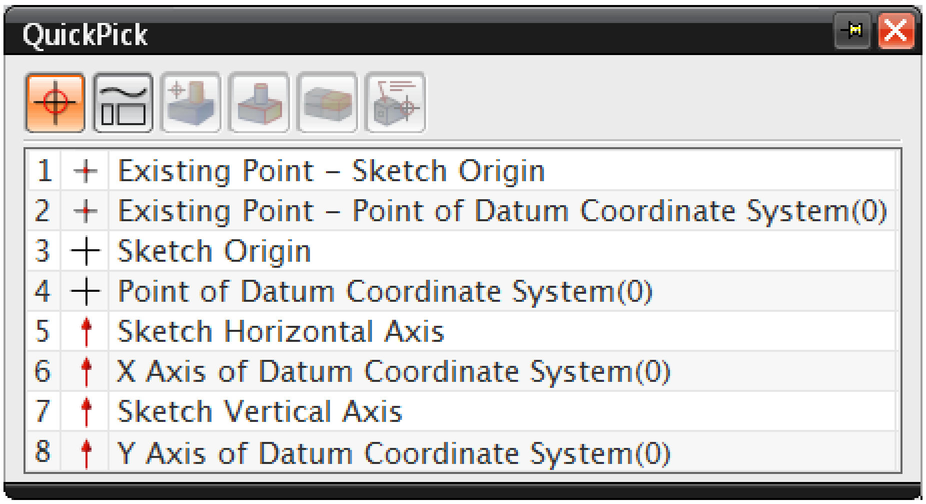
The internal sketch coordinate system objects now have the following unique names:

- **Sketch Horizontal Axis**
- **Sketch Vertical Axis**

- **Sketch Origin**, for sketches created with the **Associative Origin** option selected.

- **Point of Sketch CSYS**, for sketches created without the **Associative Origin** option selected.

These unique names differentiate the sketch CSYS from other datum CSYS that may exist in the same location. When you create sketch curves or constraints to objects on top of another datum CSYS, you can now easily select the object you want using the **QuickPick** dialog box.



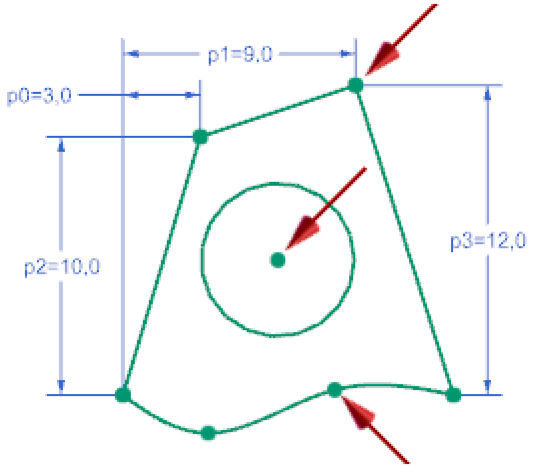
Where do I find it?

Associative Origin option

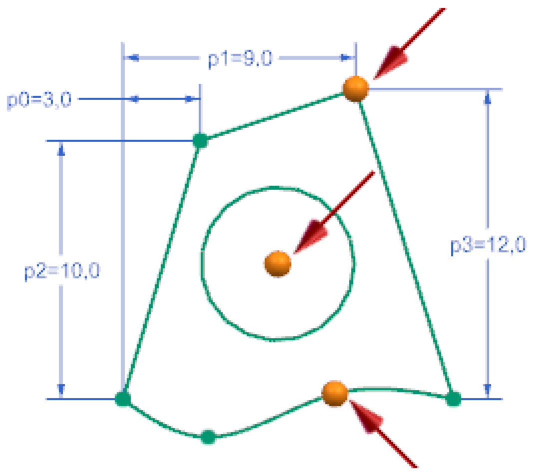
Application	Modeling, Drafting, Shape Studio, Sheet Metal
Toolbar	**Direct Sketch® Sketch**
Menu	**Insert® Sketch**
Location in dialog box	**Settings** group

Vertex display enhancement

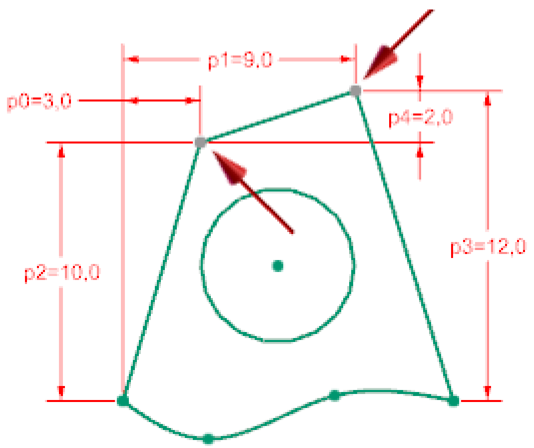
What is it?



While sketching you can now see the vertices of curves. Moving vertices is easier because there is an object that can be selected.



The display vertices now changes when you select them . This helps differentiate between dragging a curve and dragging a vertex of a curve.



Over constrained curves and vertices now highlight in your default **Unsolved Curves** color so that you can better view the overconstraint problem.

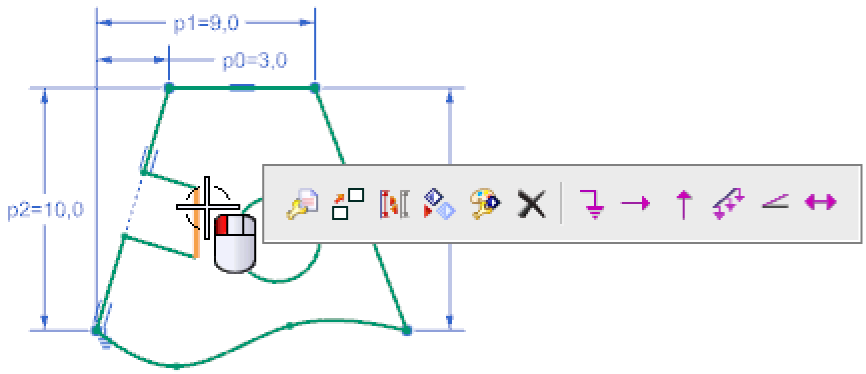
Where do I find it?

Application	Modeling, Drafting, Shape Studio, Sheet Metal
Prerequisites	In the Sketch task environment, or while a sketch is active

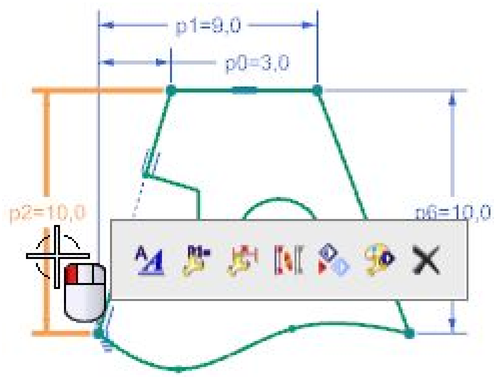
Sketch context toolbar enhancement

What is it?

When you select or right-click sketch curves a context toolbar appears next to your cursor.



When you select or right-click sketch dimension a different context toolbar appears next to your cursor.



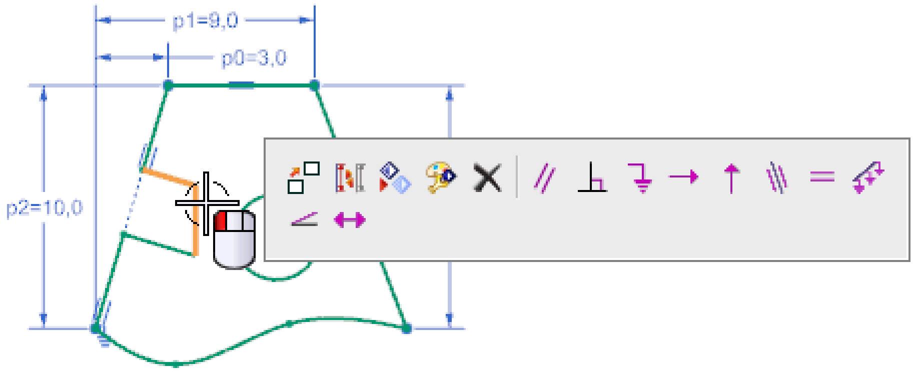
Where do I find it?

Application	Modeling, Drafting, Shape Studio, Sheet Metal
Prerequisites	Sketch task environment, or while a sketch is active

Constraints context toolbar enhancement

What is it?

You can now create sketch constraints using the context toolbar. After you select curves, the context tool bar will display all the possible constraints for the selected curves.



You can also right-click selected objects and choose a constraint from the list.

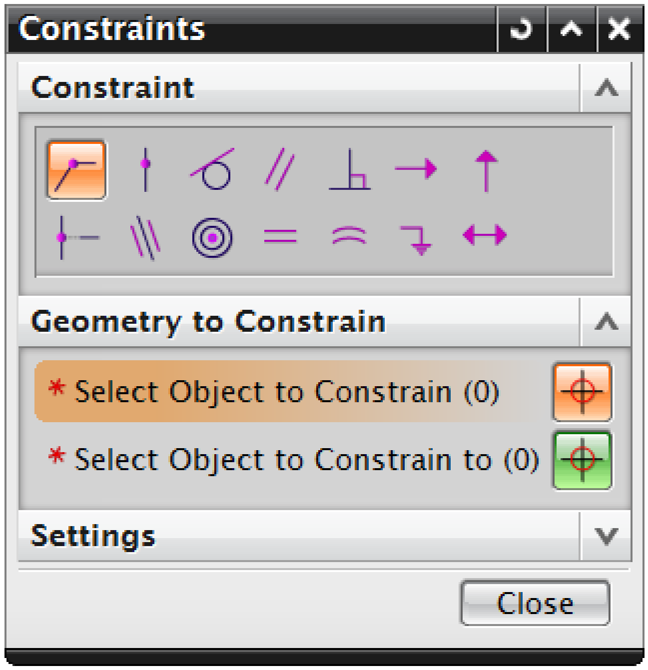
Where do I find it?

Application	Modeling, Drafting, Shape Studio, Sheet Metal
Prerequisites	Sketch task environment, or while a sketch is active

Constraints dialog box enhancement

What is it?

When you choose the **Constraints** command, NX now displays a dialog box. You must first select a constraint type and then select the objects that you want to constrain. This new workflow allows you to quickly create the same constraint on multiple objects.

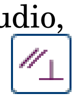


When the **Select Object to Constrain** option is active, you can select multiple objects. In the **Settings** group, you can select the constraints that you want to display by default in the **Constraints** group.

Why should I use it?

In previous releases, applying constraints to multiple objects was cumbersome and error-prone. You had to select the objects first and the constraint you could apply depended on the objects you selected. For example, if you wanted to apply a **Tangent** constraint and you selected an arc and an end point, you could only apply a **Fixed** or **Point on Curve** constraint. Now, because you must select the constraint type first, it becomes easy to select two objects, and then continue to select the next two objects.

Where do I find it?

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

Constraints quick reference

Overconstrained sketch enhancement

What is it?

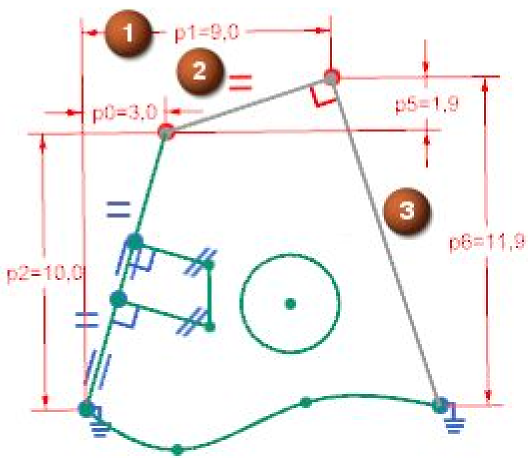
Notification of overconstrained and conflicting sketch situations has been enhanced. When a dimension or constraint is added that overconstrains a sketch, the following **Update Sketch** message is displayed;

```
The sketch geometry is overconstrained after the last edit.
```

```
The constraints and dimensions that are related to the problem are
highlighted in the Overconstrained Objects color.
```

```
The curves that are not positioned correctly are shown in the Unsolved
Curves color.
```

When you select an overconstrained dimension the status line notifies you that the dimension is overconstrained. When you place your cursor over sketch objects a tool tip also notifies you of that the object is overconstrained.



1. Overconstrained dimension is red.
2. Overconstrained constraint is red.
3. Unsolved curve is gray.

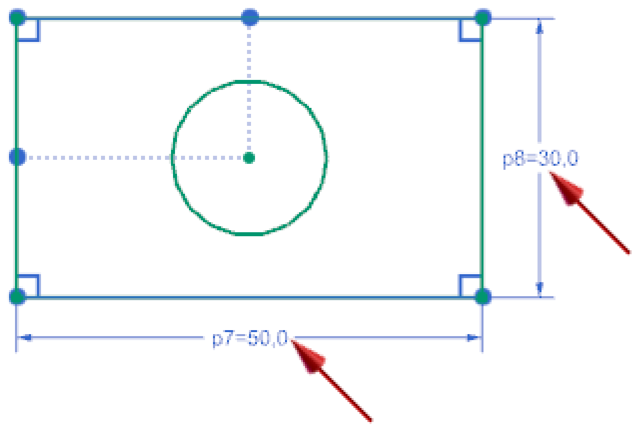
Where do I find it?

Application	Modeling, Drafting, Shape Studio, Sheet Metal
Prerequisites	Sketch task environment, or while a sketch is active

Dimension creation enhancement

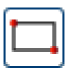
What is it?

The customer default, **Create Dimensions for Typed Values** has been changed to create driving dimensions as you enter specific values while creating sketch geometry.



Where do I find it?

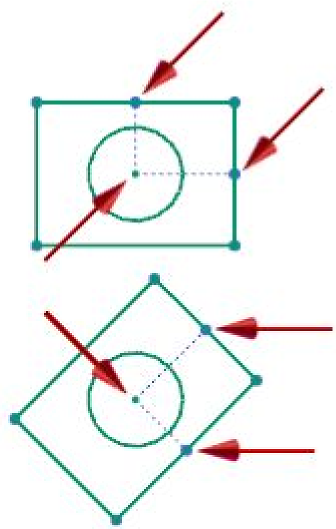
Application	Modeling, Drafting, Shape Studio, Sheet Metal
Prerequisites	Sketch task environment, or while a sketch is active when creating new geometry.
Menu	**File® Utilities® Sketch® Inferred Constraints and Dimensions® Create Dimensions for Typed Values** check box



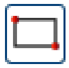
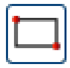
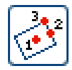
Rectangle constraints enhancement

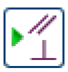
What is it?

You can now infer constraints from the mid points to the center of rectangles when you use the **From Center** option.



Where do I find it?

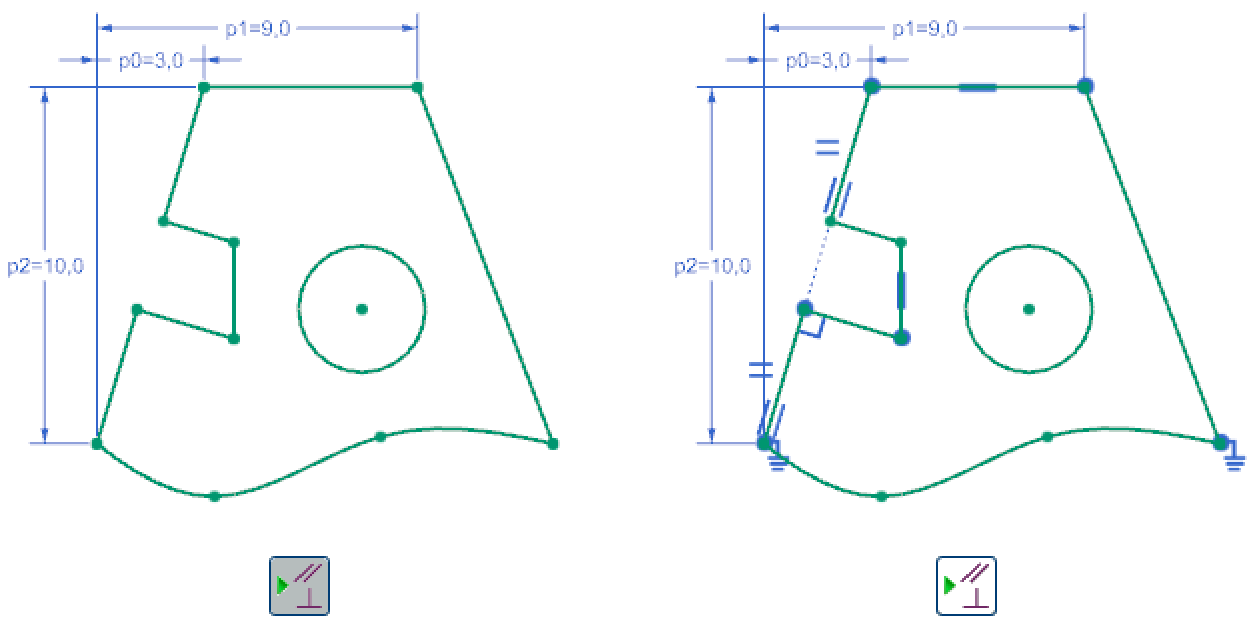
Application	Modeling, Drafting, Shape Studio, Sheet Metal
Toolbar	(Modeling, Shape Studio, Sheet Metal) **Direct Sketch® Rectangle**  (Sketch task environment and Drafting) **Sketch Tools® Rectangle** 
Menu	(Modeling, Shape Studio, Sheet Metal, Drafting) **Insert® Sketch Curve® Rectangle** (Sketch task environment) **Insert® Curve from Curves® Rectangle**
Location in dialog box	**Rectangle Method** group® **From Center** 



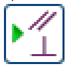
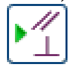
Display Sketch Constraints enhancement

What is it?

There is now a single option to turn constraint constraints on and off. The new **Display Sketch Constraints** option replaces both the **Show All Constraints** and **Show No Constraints** commands.



Where do I find it?

Toolbar	(Modeling) **Direct Sketch® Display Sketch Constraints**  (Drafting and Sketch task environment) **Sketch Tools® Display Sketch Constraints** 
Menu	**Tools® Sketch Constraints® Display Sketch Constraints**

Sketch Preferences enhancement

What is it?

Display Constraint Symbols

Sets the initial display of constraint symbols so that they are on or off.

Constraint Symbol Size

Sets the initial default size of constraint symbols. After a sketch is created you must use the **Edit® Style** command to change the size of constraint symbols.

Name Prefixes

Adds a default name when you create a sketch object. The default name **Vertex** has been changed to **Point**.

Part Settings Colors

Sets the default sketch colors.

The **Conflicting Objects** preference is now named **Conflicting Constraints**.

The **Unsolved Curves** preference is new. Use this preference to set the color for geometry that is overconstrained or has conflicting constraints.

Where do I find it?

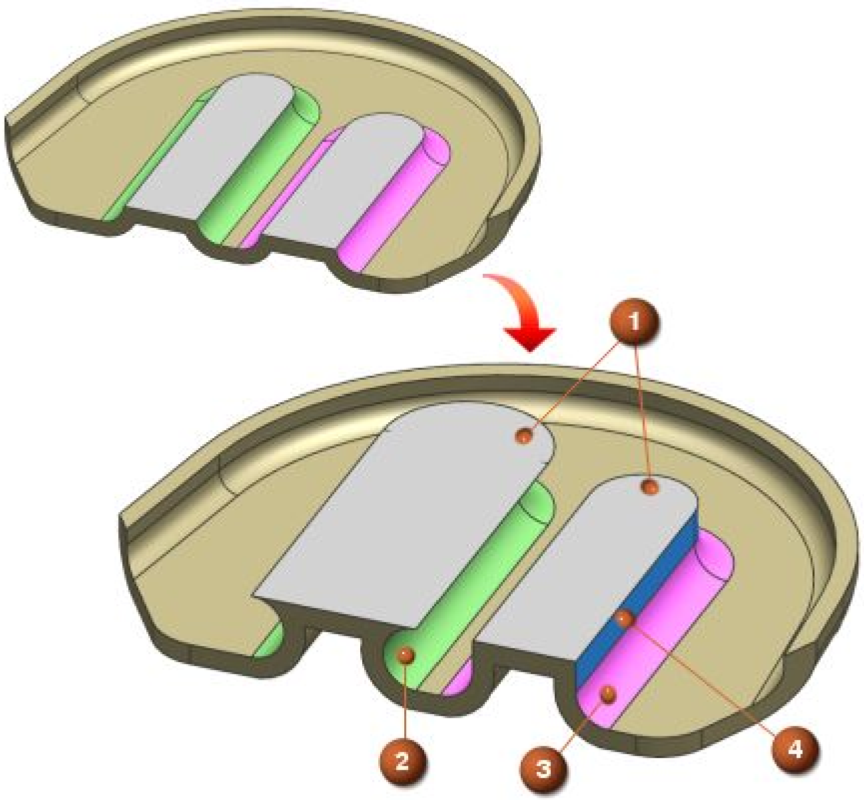
Application	Modeling, Drafting, Shape Studio, Sheet Metal
Menu	**Preferences® Sketch**
Location in dialog box	**Session Settings** tab® **Display Constraint Symbols** check box **Sketch Style** tab, **Constraint Symbol Size** **Session Settings** tab® **Name Prefixes** group® **Point** **Part Settings** tab® **Conflicting Constraints**, **Unsolved Curves**

Synchronous Modeling

Label Notch Blend

Use this command to label faces so that NX recognizes them as blends. This command is useful when you use Synchronous Modeling commands such as **Move Face** and **Resize Blend**. A labeled blend becomes a rolling ball blend.

The example shows the result of using the **Move Face** command on neighboring labeled and unlabeled faces.



1. Faces moved
2. Unlabeled faces
3. Labeled faces
4. Faces added

In History mode faces labelled as notch blend appear in the **Part Navigator** as **Label Notch Blend**.

If the model is converted to History-free mode, the label exists, but the feature does not appear in the **Part Navigator**.

Your design needs may require you to remove an existing label. To do this, you must create another notch blend feature using the same faces that you previously labeled, and select the **Delete Label** check box. A new notch blend feature is created in the **Part Navigator**.

Where do I find it?

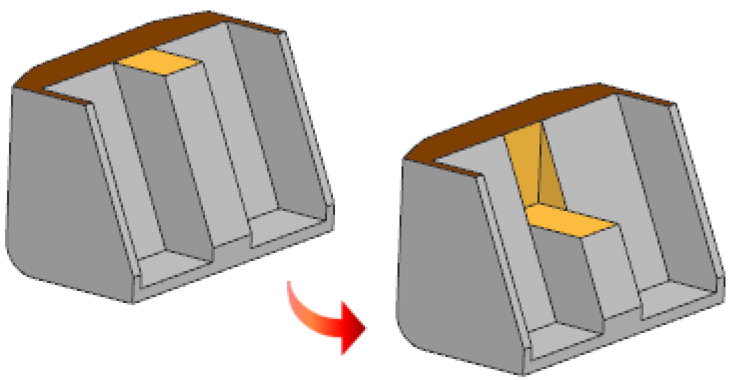
Application	Modeling, Shape Studio. Advanced Simulation, Manufacturing
Toolbar	**Synchronous Modeling® Label Notch Blend**
Menu	**Insert® Synchronous Modeling® Detail Feature® Label Notch Blend**

Move Face — Step Face enhancement

What is it?

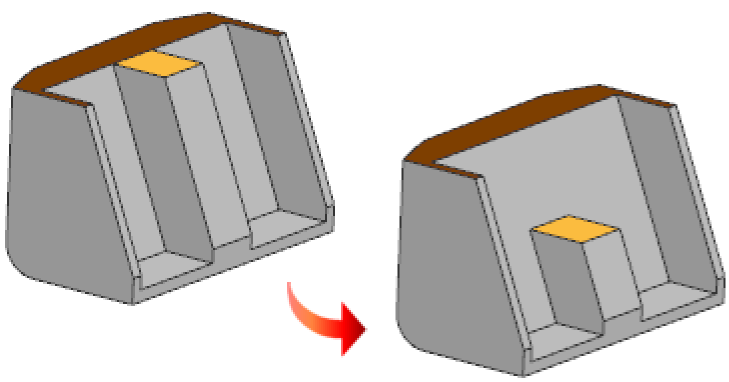
When you use the **Move Face** command, you now have more control over the extension of neighboring faces. A new **Step Face** list is added to the **Settings** group. If you select the **Extend Neighbors at Smooth Edge** option, you can extend and combine neighboring coplanar faces.

In this example the top face has been split into two faces.



Overflow Behavior = **Extend Cap Face**

Step Face = **None**



Overflow Behavior = **Extend Cap Face**

Step Face = **Extend Neighbors at Smooth Edge**

Where do I find it?

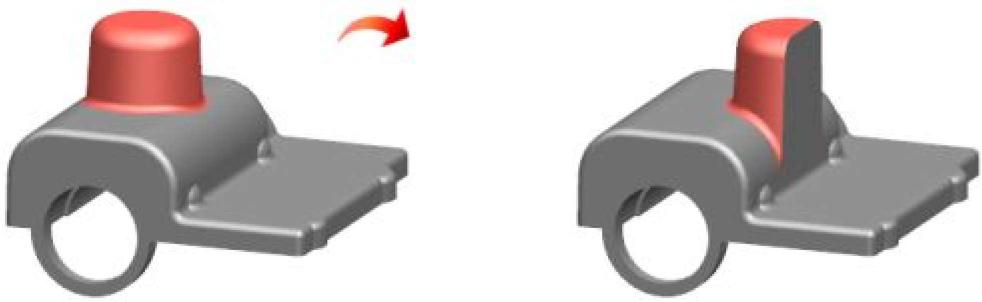
Application	Modeling, Shape Studio. Advanced Simulation, Manufacturing
Prerequisite	In the **Move Behavior** list, you must select **Move and Adapt**
Toolbar	**Synchronous Modeling® Move Face** 
Menu	**Insert® Synchronous Modeling→Move Face**
Location in dialog box	**Settings** group® **Face Change Step** list

Move Face Cut and Heal enhancement

When you use the **Move Face** command, a new **Move Behavior** list offers better control over the move behavior. You can:

- Cut a set of faces on a body using the new **Cut and Paste** option.
- Paste the cut faces immediately. To do this, you must select the new **Paste** check box. If you do not paste the faces immediately, they are still available for paste using the **Paste Face** command.
- Choose whether you want the neighboring faces to be healed. A new **Heal** check box controls healing.

In previous releases, you could not use the **Move Face** command to move a face when NX was unable to heal the neighboring faces. If you used the **Move and Adapt** option to move the boss shown, the motion face was consumed.



Note The **Move Behavior** options are available in the **Move Face** dialog box only when you create a feature.

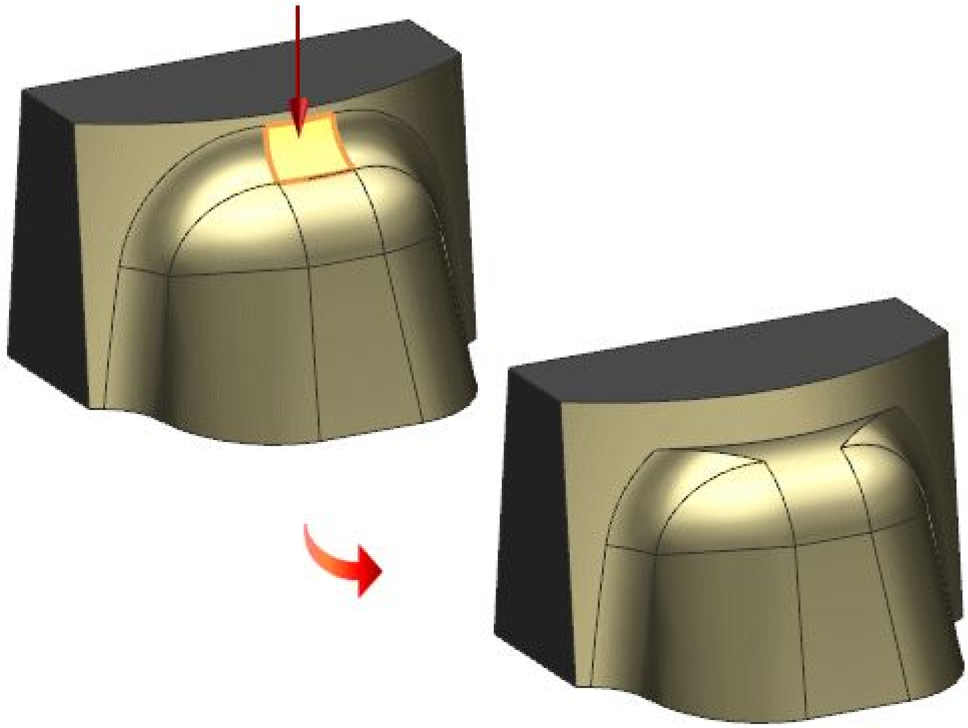
Where do I find it?

Application	Modeling, Shape Studio. Advanced Simulation, Manufacturing
Toolbar	**Synchronous Modeling**® **Move Face** 
Menu	**Insert**® **Synchronous Modeling→Move Face**
Location in dialog box	**Settings** group® **Move Behavior** list

Delete Face enhancement

You can now use the **Delete Face** command to delete a portion of a blend and cap the end of the blend where it neighbors another blend. When you use the new **Delete Partial Blend** option all faces selected must be blends.

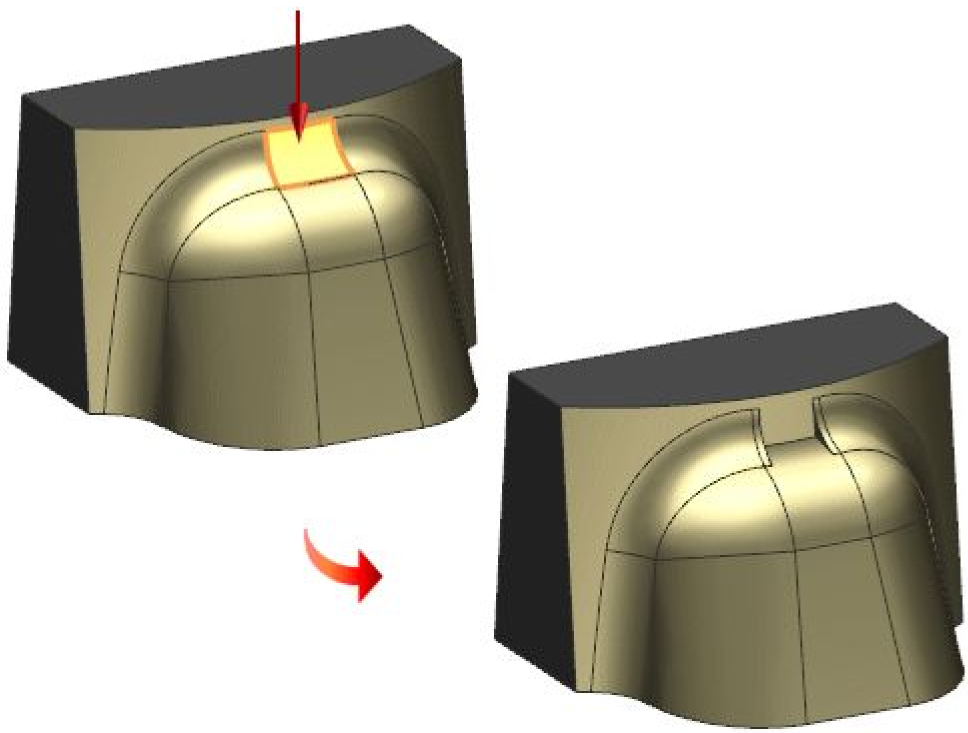
Delete Partial Blend



Delete Partial Blend

Setback = Selected Blend

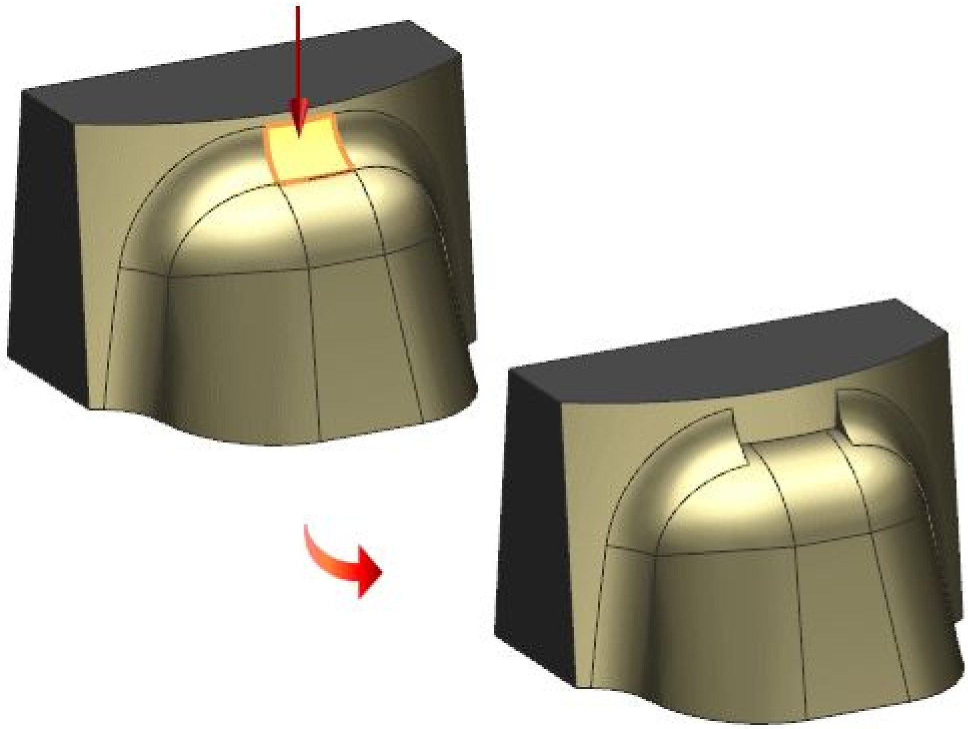
This option caps the remaining blends by a setback along the selected blend.



Delete Partial Blend

Setback = **Neighbor Blend**

This option caps the remaining blends by a setback along the neighbor of the selected blend.



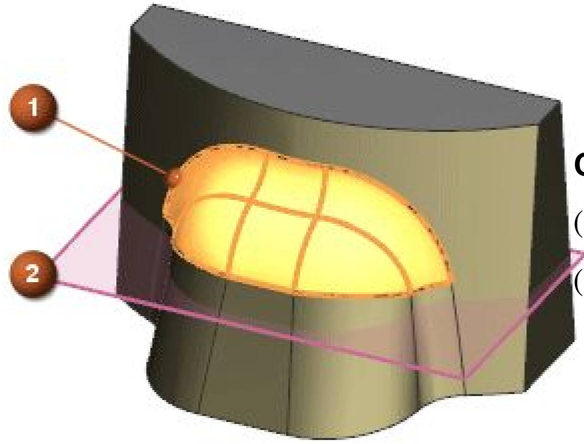
Where do I find it?

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

Delete Face with explicit cap enhancement

What is it?

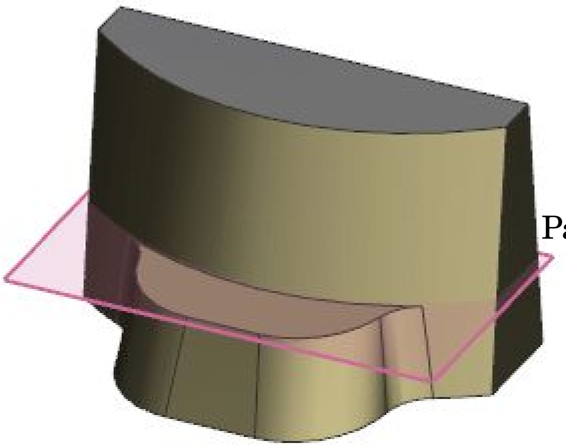
When you use the **Delete Face** command with the **Heal** option selected, you can now specify an explicit cap face. A cap face can improve healing when the remaining faces cannot close the area left after deleting the selected faces. You can select a face of the target body, the face of another body, or a datum plane as a cap face.



Cap Option = **Face or Plane**

(1) Faces to be deleted

(2) Cap face



Part with deleted faces

Where do I find it?

Application	Modeling, Shape Studio. Advanced Simulation, Manufacturing
Prerequisite	In the **Settings** group, the **Heal** check box must be selected.
Toolbar	**Synchronous Modeling**® **Delete Face**
Menu	**Insert**® **Synchronous Modeling**→**Delete Face**
Location in dialog box	**Cap Face** group® **Cap Option** list

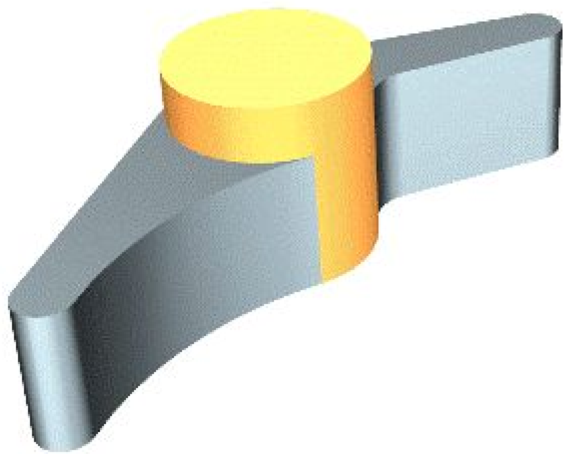
Improved problem area detection

What is it?

In synchronous modeling, there is now improved recognition of problem area conditions that are likely to result in potential errors when moving or deleting faces.

The **Alerts** messages now provide improved clues to identify the error condition and offer potential solutions.

In the example, the faces on the center boss feature are to be deleted using the **Delete Face** command. An **Alerts** message informs you that the remaining faces cannot adapt to close the area left by deleting the selected faces. Some possible solutions are given, such as using the cap option or deleting the blends first.



Modeling

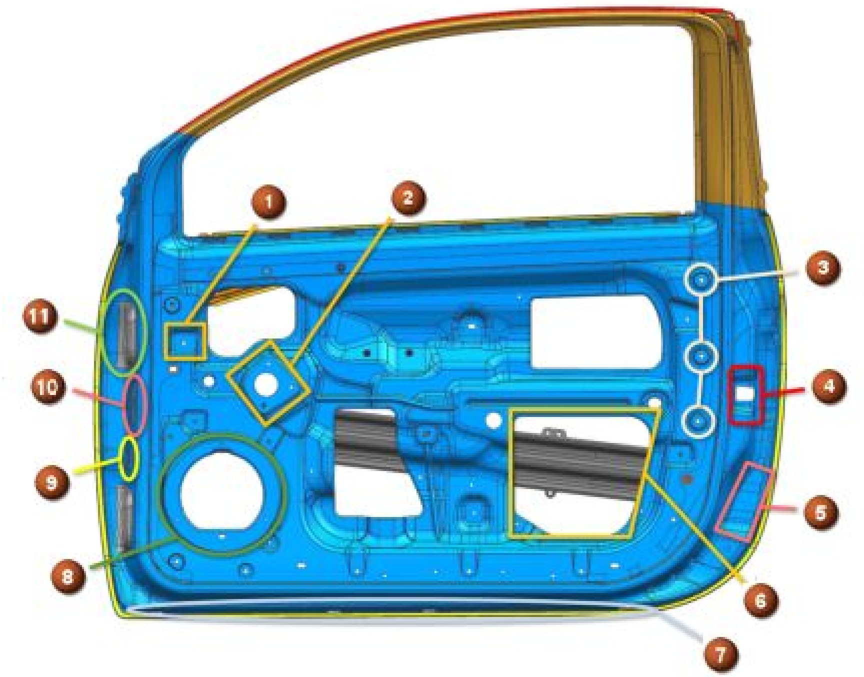
Part Module

What is it?

The Part Module suite of commands leverages modular design principles in NX, incorporating functional partitioning, modular interfaces, and ease of change.

Part Modules enable complex designs to be divided into logical portions within a part file, primarily so it can be broken out for completion by other designers. This isolated nature makes the design elements within a part file more robust, because they can be changed with minimal or no effect on the rest of the part file. A Part Module's isolated nature also makes it easy to reuse in other part files, or as the basis for new part files.

Examples of partitioned functional areas in a part file



1 Handle	2 Window	3 Interior Trim	4 Latch
5 Structural Support	6 Side Impact Block	7 Drain Holes	8 Speaker
9 Routing	10 Check Strap	11 Hinge	

At any time, the lead designer can update the Part Modules in a part file to view the progress of the features created by the assigned team members, and ensure that continuing downstream work is based on the latest geometry.

Part Modules also let you isolate areas of a large part file, making it easier to both display and update the areas.

Part Modules in the Part Navigator

Part Modules appear in the **Part Navigator** as features divided into three step-nodes:

- **Inputs** — This step-node contains extractions of the geometry objects and expressions that you select from the part at the time you create the Part Module. These objects are the parent references for the features you create later and that appear in the **Work** step-node. **Input** is the only step-node with anything in it when you first create a Part Module.

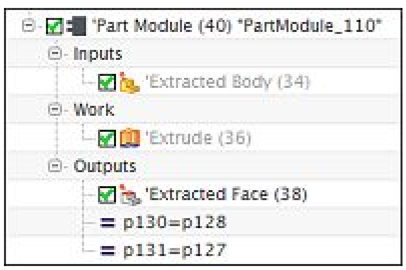
- **Work** — This step-node is where all new features you create in a Part Module are collected and displayed. Features you create can be based only on the extracted geometry and expressions in the **Input** step-node.

- **Outputs** — This step-node contains extractions of those features and expressions you specify from the **Work** step-node. Downstream features of the main part are based on this output. When a Part Module is deactivated, its **Outputs** are what the rest of the part sees as the geometry of the Part Module feature.

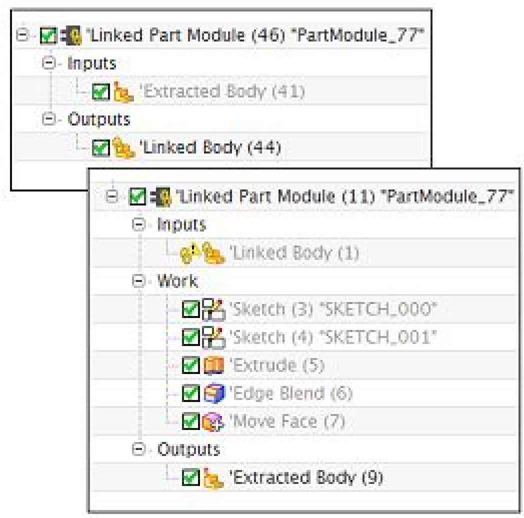
Part Modules and Linked Part Modules

A Part Module exists either as a **Part Module** solely within the main part file, or as a **Linked Part Module** that links the main part to a second part using an interpart reference (WAVE). The Linked Part Modules exist in different part files.

Part Module



Linked Part Module



In a Linked Part Module:

- Objects in the **Inputs** step-node are copied and WAVE Linked from the main part file to the separate link part file.

- Features in the **Work** step-node are moved to the linked part file.

- Objects in the **Outputs** step-node are copied and WAVE Linked from the separate link part back to the main part file.

- Associativity is preserved between the main part file and the linked part file.

- After all work is completed, you can merge the linked part into the main part. The main part appears as if all the design work was performed there.

Why should I use it?

The Part Module functionality simplifies the creation and editing of complex parts. It enables parallel work within a single part, and efficient model updates. It also supports reuse.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Feature Replay**
Menu	**Format® Part Module** The menu contains additional commands that are not available on the toolbar.

Emboss Body

What is it?

The new**Emboss Body** command replaces the **Emboss Sheet** command.

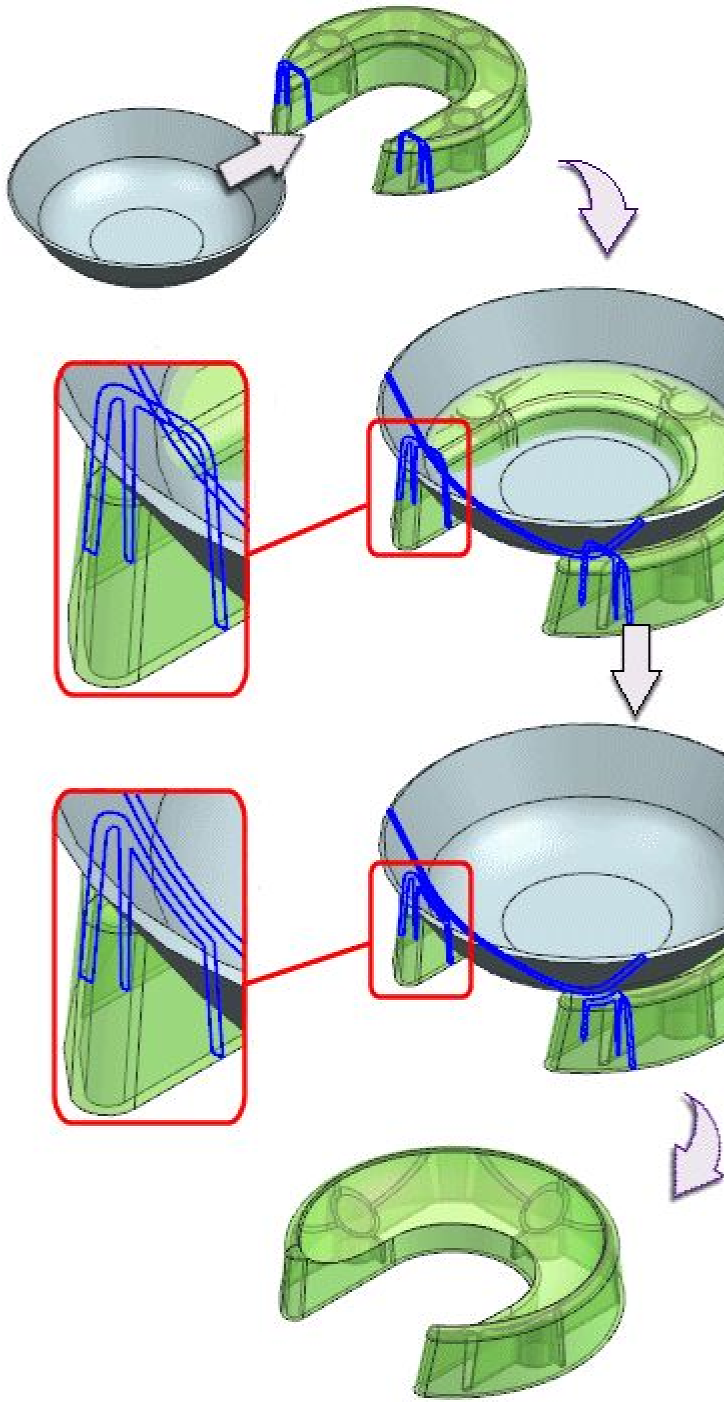
Use it to modify a solid or sheet body by combining a region of faces from another intersecting solid or sheet body.

You can:

- Emboss a solid body using another solid body or a sheet body.
- Emboss a sheet body using another sheet body or a solid body.

You can also:

- Specify a clearance between the tool and the target bodies.
- Specify a thickness when the target is a solid body.
- Reverse the material side when the target is a sheet body.
- Reverse the emboss direction when the target is a sheet body.
- Specify a region to emboss when there are multiple possibilities.



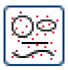
Example of a thin wall solid body embossed by another thin wall solid body

Why should I use it?

Use the **Emboss Body** command when you need to modify a sheet or solid body so it avoids interference with another body.

Where do I find it?

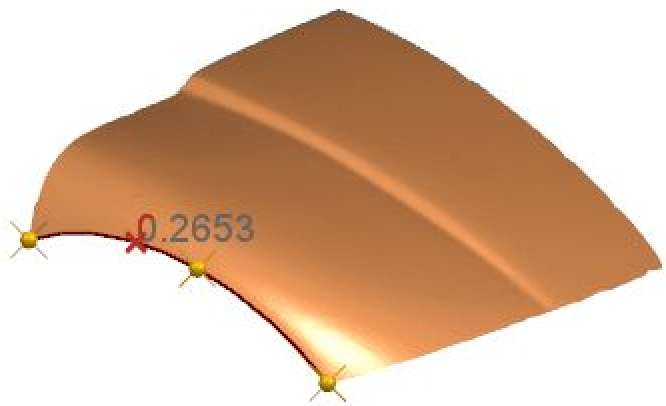
Application	Modeling and Shape Studio
Menu	**Insert® Combine® Emboss Body**



Fit Curve

What is it?

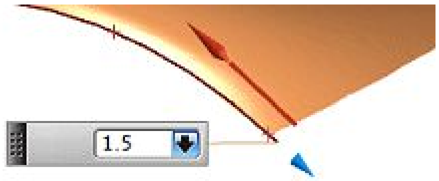
Use the new **Fit Curves** command to create a quadric curve or a spline by fitting it to specified data points.



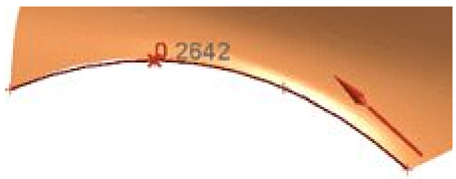
Example of fitting a spline on a facet body

You can fit splines, lines, circles, and ellipses.

- Splines, circles and ellipses, can be open or closed.
- Lines and open circles, splines, and ellipses can be extended using on-screen handles or input boxes.



- The maximum fitting error and the location of the error is displayed on-screen.



The data points can reside in a set of chained points, or on facet bodies, curves, or faces.

For splines, you can set endpoint and inner continuity constraints, and you can control the accuracy and shape of the fit by specifying:

- **Degrees and Segments**
- **Degree and Tolerance**
- **Template Curve**

An associative **Fit Curve** feature is created when curves, faces, chained points, or selected points are the target. Any modifications to the target object causes the associated fit curve to update accordingly. Curves fitted to facet bodies have no associativity.

Why should I use it?

Fitting curves as cross sections, to either scanned point cloud data or facet bodies, creates the necessary guide and section curves to begin surfacing, which is a fundamental part of Reverse Engineering workflows.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Curve® Fit Curve** **Shape Studio® Curve Drop-down® Fit Curve**
Menu	**Insert® Curve® Fit Curve**

Advanced Curve Fit

What is it?

When you use the **Section Curve** command, you can now control advanced parameterization of the output curves using the **Advanced Curve Fit** option. This option replaces the **Curve Fit** option that was available in previous releases.

When you select the **Advanced Curve Fit** check box, you can select one of the following options from the **Method** list.

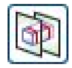
- **Degree and Segments**: Use this option when you want explicit control on the parameterization of output curves.

- **Degree and Tolerance**: Use this option when you want to use the maximum specified degree to achieve the specified tolerance value.

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

Note If you do not select the **Advanced Curve Fit** check box, NX automatically fits the resulting curve to a Quintic curve (order 5 curve) based on the specified tolerance value.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Curve® Section Curve** 
Menu	**Insert® Curve from Bodies® Section**
Location in dialog box	**Settings** group

Advanced Curve Fit for Bridge Curve and Curve on Surface

What is it?

The **Advanced Curve Fit** option has been added to the **Bridge Curve** and **Curve on Surface** 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 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, and 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.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Curve® Curve on Surface** **Curve® Curve from Curves® Bridge Curve**
Menu	**Insert® Curve® Curve on Surface** **Insert® Curve from Curves® Bridge Curve**
Location in dialog box	**Settings** group

General Conic enhancements

What is it?

The **General Conic** command has been upgraded with a modern NX dialog box and the following functional enhancements:

- Full feature support, so conics are associative to their parent geometry.
- You can edit the defining points of conics.
- You can change the conic type during edit.
- You can change the conic curve length during creation and edit using handles.

Why should I use it?

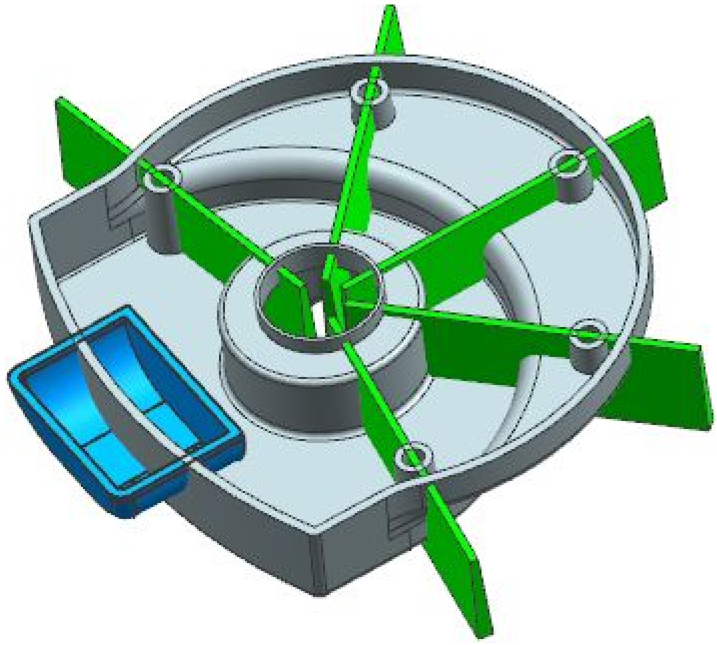
General Conic now supports feature based modeling in an easy to use updated user interface.

Where do I find it?

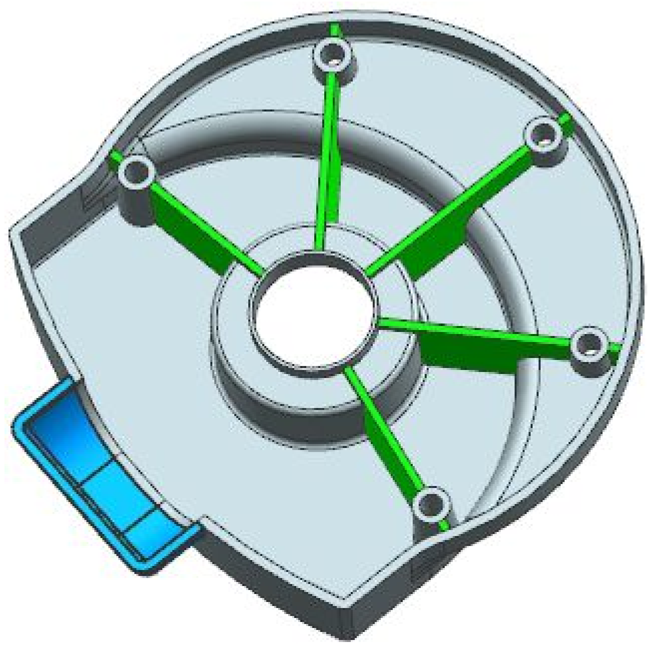
Application	Modeling
Toolbar	**Curve® General Conic**
Menu	**Insert® Curve® General Conic**

Unite with region selection

When uniting solid bodies that have overlapping, or excess material, you can now use region selection to identify a collection of faces that should be kept or removed when the solid bodies are united.



Before unite



After unite with region selection

Why should I use it?

Region selection saves you from having to trim the bodies before uniting them. This simplifies the part navigator feature tree, as there are no separate trim body features listed.

Where do I find it?

Application	Modeling
Toolbar	**Feature→Combine Drop-down→Unite**
Menu	**Insert→Combine→Unite**
Location in dialog box	**Region** group

Pattern Feature enhancements

What is it?

Pattern Feature has the following improvements:

- You can change the **Pattern Method** when editing a pattern feature. For example, you could change the **Pattern Method** from **Variational** to **Simple**, and vice versa.

  If for some reason the **Pattern Method** cannot be changed, an alert message appears. This could happen, for example, when the pattern feature being edited is already the input for another pattern feature.

- You can clock instances of a **Pattern Feature** from the **Part Navigator** or from a scene by right-clicking an instance and choosing the new **Clock** option.

- **Pattern Feature** now supports additional feature types.

Where do I find it?

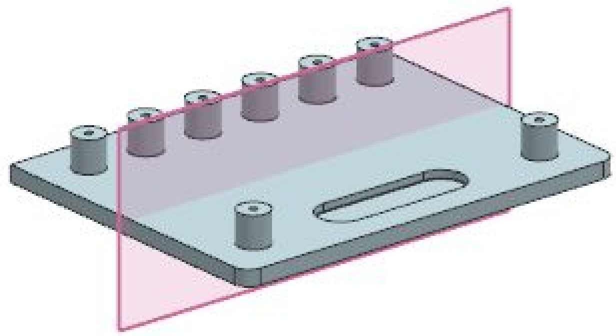
Application	Modeling and Shape Studio
Prerequisite	History mode
Toolbar	**Feature® Pattern Feature**
Menu	**Insert® Associative Copy® Pattern Feature**

Mirror Feature

What is it?

The functionality for mirroring features is improved. You can mirror features that could not be mirrored with the previous **Mirror Feature** command. These include curve features, datum features, and pattern features created with the **Pattern Feature** command that was introduced in NX 8.0.

When you mirror one or more features, a single mirror feature is created.



The updated **Mirror Feature** command includes the following capabilities:

- You can mirror existing features except for legacy instance arrays, legacy patterns, and legacy mirror features.
- You can specify whether thread features and helix features maintain their original handedness when mirrored.
- When the mirrored features include a coordinate system, you must specify which two axes are mirrored. NX derives the third axis to ensure that the coordinate system is right-handed.

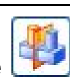
The **Mirror Feature** dialog box is changed to accommodate the enhancements. The pre-NX 8.0.1 **Mirror Feature** dialog box is available only when you edit a legacy mirror feature.

Why should I use it?

The **Mirror Feature** command is useful in the following cases:

- When you want features that are symmetrical to other features.
- When you want a similar feature at a comparable location on the other side of the mirror plane, but do not need absolute symmetry.

Where do I find it?

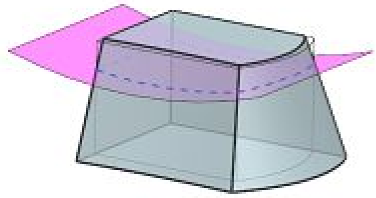
Application	Modeling and Shape Studio
Toolbar	**Feature® Mirror Feature** 
Menu	**Insert® Associative Copy® Mirror Feature**

Draft enhancements

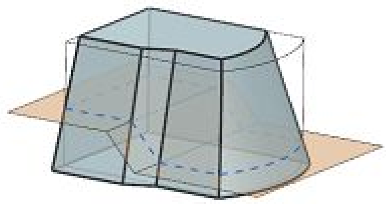
What is it?

When you add draft to a part you can now specify Draft from:

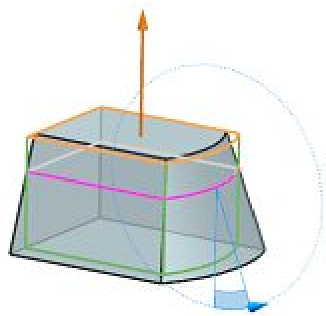
- A non-planar surface



- Multiple faces

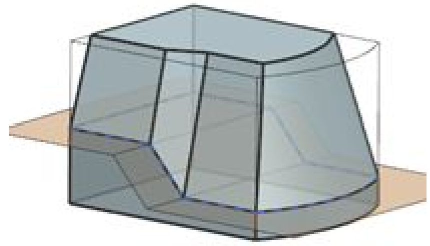


- A planar face or plane



- Both sides of a parting surface

- One side if a parting surface



Where do I find it?

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

New Selection Intent rules for Draft and Edge Blend

What is it?

Two new Selection Intent curve rules make it easier to draft and blend face group perimeters and rib-like structures in a part model.

Outer Edges of Faces

This curve rule lets you easily collect the perimeter edges of a group of faces by selecting the faces either directly or by selecting the features that created, modified, or collected those faces. Edges that are common to the faces selected under this rule are removed; that is, all of the selected faces are treated as if they were one face, and the perimeter edges of that one face are selected. Note that this rule always collects all of the edges of the faces selected.

Rib Top Face Edges

This is essentially the **Outer Edges of Faces** rule with additional edge removal, which simplifies the drafting or blending of ribs or rib-like structures. Under this rule, only perimeter edges of the same convexity are ultimately collected for the selected faces.

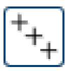
Why should I use it?

Use **Outer Edges of Faces** when you want to select all of the perimeter edges of selected faces, and you want to be able to select them by selecting the feature that created them or the group that contains them.

Use **Rib Top Face Edges** when you want to apply a **Draft** or **Edge Blend** to ribs or rib-like structures.

Where do I find it?

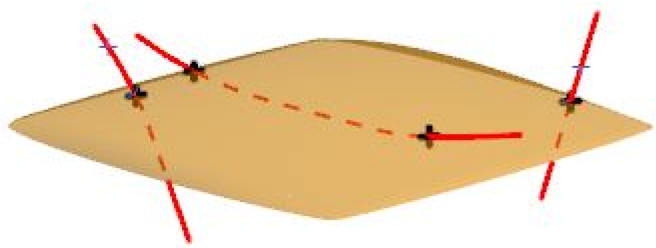
Application	Modeling, Shape Studio, and NX Sheet Metal
Prerequisite	Available only with the **Edge Blend** and **Draft** commands. (With the **Draft** command, **Type** must be set to **From Edges**.)
Toolbar	**Selection Bar® Selection Intent** pull-down menu
Location in dialog box	**Edge Blend** dialog box® **Edge to Blend** group® **Select Edge** **Draft** dialog box® **Stationary Edges** group® **Select Edge**



Point Set enhancement

What is it?

You can use the new **Point Set type→Intersection Point** to create a set of points that can be used in conjunction with **Isolate Object of Feature**, or any time you need a set of points as a feature.



Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Feature→Datum/Point drop-down menu→Point Set** 
Menu	**Insert→Datum/Point→Point Set**
Location in dialog box	**Type** list® **Intersection Points**

Isolate Object of Feature

What is it?

Use the **Isolate Object of Feature** command to control which individual object of a feature is used for downstream features. The individual object can be a body, a curve chain (one or more curves), or a point. It is determined by its proximity to a selected plane or point.

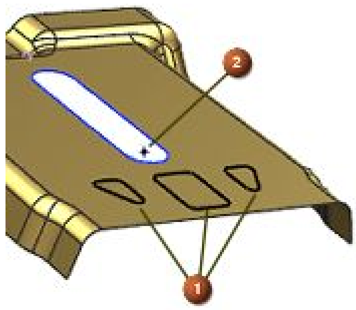


Figure. Sketch with 3 chains of curves (1) & Proximity point for Isolate feature (2)

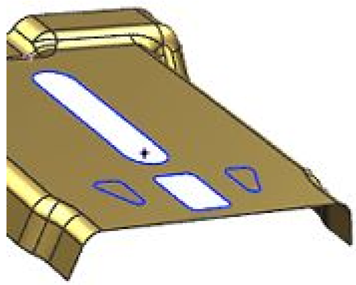


Figure. Extrusion created using the Isolate feature

Figure. Slot (with Proximity point) moved

Where do I find it?

Application	Modeling and Shape Studio
Menu	**Insert® Associative Copy® Isolate Object of Feature**

Delete Body

What is it?

Use **Delete Body** to delete one or more bodies associatively. When you execute the command and select the bodies to delete, a **Delete Body** feature is created. You can undo the deletion later simply by suppressing or deleting the **Delete Body** feature.

You can select exceptions when you create the **Delete Body**, which will be the only features output by the original selected feature.

Why should I use it?

Use **Delete Body** when you want to ensure certain bodies are not referenced downstream, or when a feature outputs more bodies than you actually need in your design.

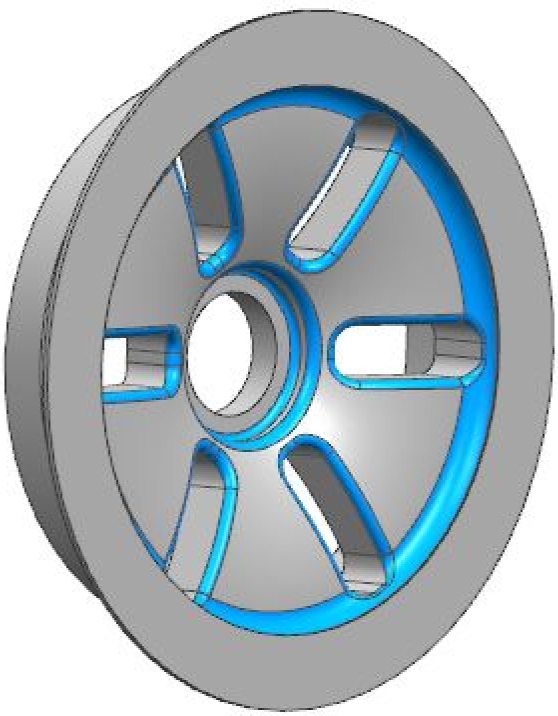
Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Feature® Delete Body**
Menu	**Insert® Trim® Delete Body**

Assign Feature Color

What is it?

Use **Assign Feature Color** to assign a color property to a feature. The color is inherited by all faces that are created or modified by the feature. When you modify a face several times, it takes on the color of the last feature you used to modify it.



Color property assigned to edge blends in a wheel

Why should I use it?

This enhancement lets you view different features and their output faces in a specific color.

Where do I find it?

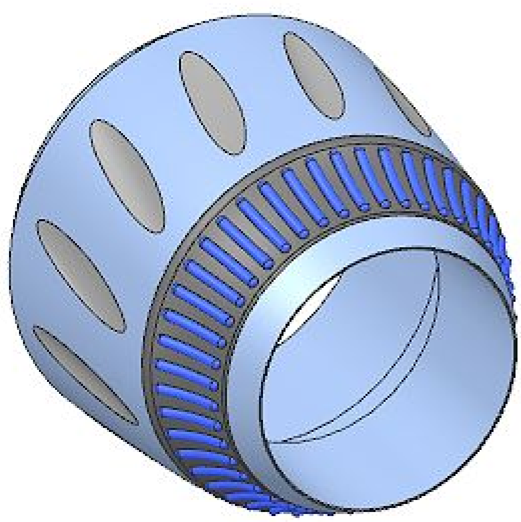
Application	Modeling, Shape Studio, NX Sheet Metal
Toolbar	**Edit Feature® Assign Feature Color**
Menu	**Edit® Feature® Assign Feature Color**
Graphics window	Right-click a highlighted feature and choose **Assign Feature Color**.

Assign Feature Group Color

What is it?

Use **Assign Feature Group Color** to add a color property to one of the following categories in an embedding **Feature Group**:

- The color is added to the feature group and any new features that are added to the group.
- The color is added to the feature group and all current members of the group.
- The color is added to the feature group and the first level of features in the group.



Various colors assigned to different feature groups

Why should I use it?

Use this enhancement to view different features or feature groups in specific colors.

Where do I find it?

Application	Modeling, Shape Studio, NX Sheet Metal
Prerequisite	On creation, the Feature Group must have the **Embed Feature Group Members** option enabled.
Toolbar	**Edit Feature® Assign Feature Group Color** 
Menu	**Edit® Feature® Assign Feature Group Color**
Part Navigator	Right-click a feature group and choose **Assign Feature Group Color**.

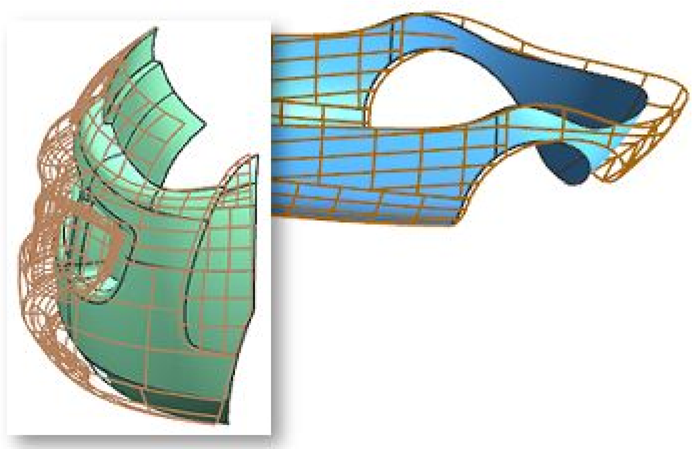
Global Shaping enhancements

What is it?

Global Shaping by Function is now named **Global Shaping**, and has two new Types:

- **By Surface**
- **By Curve**

All three **Global Shaping by Surface** types are replaced by the new **Global Shaping→By Surface**.



Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Edit Surface**
Menu	**Edit→Surface→Global Shaping**

Helix enhancements

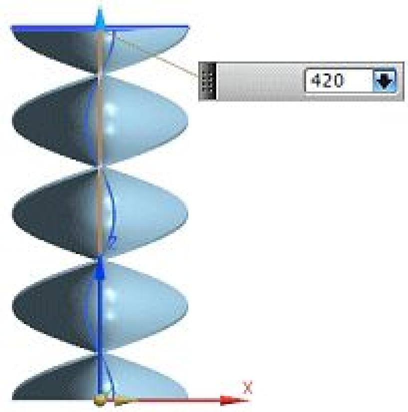
What is it?

The **Helix** command workflow is enhanced in a number of ways. You can now:

- Create a helix with an associative start point and direction.
- Use law types to define variable pitch and size values.
    - Constant
    - Linear
    - Cubic
    - Linear along Spine
    - Cubic along Spine
    - By Equation
    - By Law Curve

Helical springs of variable radius and pitch using law functions

- Control the length of a helix by defining start and end limits using on-screen handles, or by number of turns.



- Define a helix along a 3D path.

Why should I use it?

Use the **Helix** command when a helical driven object is required. Helical-driven designs are common in such products as conveyor systems, drilling equipment, tools, as well as in industrial design styling applications.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Curve® Curve Drop-down® Helix**
Menu	**Insert® Curve® Helix**

Expression formula locking and filtering enhancements

Expression formula locking enhancements

You can now lock and unlock one or more expression formulas in the **Expressions** dialog box using the new **Lock Formula** option. The expression values can still update while the formulas are locked.

Expressions with locked formulas are grayed out and display the lock icon .

Caution An overriding interpart expression can override the value of an expression, ignoring the Lock Formula.

Note To unlock a locked expression, right-click the expression and choose **Unlock Formula**.

Expression filtering enhancements

- You can now use the **Object Parameters** option to filter for the following types of expressions associated with features:
  - String feature expressions
  - Vector feature expressions
  - Point feature expressions
  - Boolean feature expressions
  - Suppression controlling expressions
- You can now use the **Filter by Feature Type** option to filter for feature expressions of a selected feature type in a model.

The **Filter by Type** category is now called **Filter by Expression Type**.

Where do I find it?

Application	Modeling and Assemblies
Menu	**Tools→Expression**
Location in dialog box	Locking or unlocking: Right-click an expression® **Lock Formula** or **Unlock Formula** Filtering: **Listed Expressions** group® Categories list

Create Multiple Interpart Expressions

What is it?

Use the new **Create Multiple Interpart Expressions**  option to simultaneously link multiple expressions from a source part into the current work part.

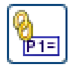
You can still create a single interpart expression using the new **Create Single Interpart Expression**  option. To edit interpart expressions you can use the new **Edit Multiple Interpart Expressions**  option.

Naming rule options

You can choose from several new automated naming options when you use **Create Multiple Interpart Expressions**.

- **Add Prefix/Add Suffix** adds a prefix or suffix to the source expression name when the interpart expression is created.
- **Replace** lets you specify a name to find and replace with another name of your choosing.
- **Replace with Index** lets you replace the source expression name with a name of your choosing. As more interpart expressions are created an index is added to the name.

Where do I find it?

Application	Modeling and Assemblies
Menu	**Tools→Expression**
Location in dialog box	**Create Single Interpart Expression**  **Create Multiple Interpart Expressions**  **Edit Multiple Interpart Expressions** 

G2 (Curvature) tolerance

What is it?

You can now set a default value for the **G2 (Curvature)** tolerance in the **Customer Defaults** dialog box. The **G2 (Curvature)** tolerance value controls the threshold for curvature continuity of freeform features.

In previous releases, you could set default values only for **Distance** (G0) and **Angle** (G1) tolerances.

Where do I find it?

Application	Modeling and Shape Studio
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Modeling® General® Tolerance/Scales** tab

Extract Geometry

What is it?

The **Extract Body** command is now called **Extract Geometry** and has been enhanced.

The **Type** of geometry you can extract has been augmented, and some types have been relocated to the Extract Geometry dialog box from the Associative Copy menu:

- **Composite Curve** (was Associative Copy® Composite Curve)
- **Point** (new)
- **Datum** (new) Allows extracting copies of Datum Axes, Datum Planes, and Datum Coordinate Systems.
- **Mirror Body** (was Associative Copy® Mirror Body)

There is now an **Associative** check box, allowing the creation of non-associative copies of geometry.

Why should I use it?

Use the **Extract Body** command to create an associative or non-associative copy of existing geometry or datums.

Where do I find it?

Application	Modeling and Shape Studio
Menu	**Insert® Associative Copy® Extract Geometry**
Location in dialog box	**Type** list and **Settings**

Positive (solid) Holes

What is it?

You can create “positive” holes without specifying target bodies for them. This can be useful when the target body has not yet been specified or is not yet present in the part, but your hole templates are ready.

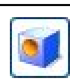
Positive holes are solid bodies, and are created from the **Hole** dialog box using **None** for the Boolean and **Along Vector** for the hole direction.

Positive holes can include symbolic (internal) threads, which will later update correctly when a target solid body is available and specified for the hole, and the Boolean is changed to **Subtract**.

Why should I use it?

Use this capability to create holes and threads in a part, even before the intended target body has been specified. It is also useful when you are migrating data from other CAD systems using CMM and you need to include positive type holes in the conversion.

Where do I find it?

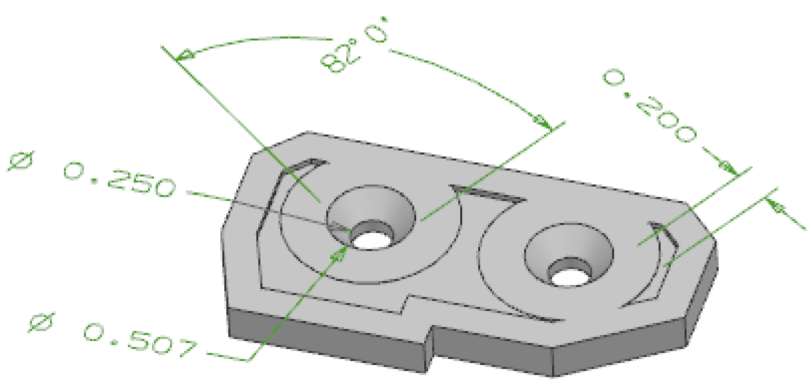
Application	Modeling, Shape Studio, and NX Sheet Metal
Toolbar	**Feature® Hole** 
Menu	**Insert® Design Feature® Hole**
Location in dialog box	**Direction** group® **Hole Direction** option set to **Along Vector** **Boolean** group® **Boolean** option set to **None**

Feature Dimensions available for more features types

What is it?

Feature Dimensions are now available for these additional feature types:

- **Edge Blend**
- **Shell**
- **Chamfer**
- **Revolve**
- The following types of **Hole** features:
  - o **General Hole**
  - o **Drill Size Hole**
  - o **Screw Clearance Hole**
  - o **Symbolic Thread**



Feature Dimensions for countersunk holes and shell

The **Feature Dimension** command lets you edit the value of any expression displayed as a feature dimension or PMI Feature dimension, without your needing to edit the feature itself.

Where do I find it?

Application	Modeling, Shape Studio, NX Sheet Metal
Toolbar	**Edit Feature® Feature Dimension** 
Menu	**Edit® Feature® Feature Dimension**
Graphics window and Part Navigator	(View feature parameters as dimensions) Right-click and choose **Show Dimensions** on a supported feature. (View sketch driving dimensions) Right-click and choose **Show Dimensions** on: • Features with internal sketches • Sketches

Automatic reordering of feature groups

What is it?

You can now quickly create both timestamped and floating feature groups in the same part. You can also control whether features are reordered when a new feature is created.

Why should I use it?

You may want to organize your part to have a certain kind of feature (such as a tool body feature) in a feature group always appear at the bottom of the Part Navigator. Yet you also want other feature groups that are positioned at the timestamp at which they were created. This can be accomplished by using a mix of timestamped and floating feature groups in the same part.

Where do I find it?

Application	Modeling and NX Sheet Metal
Prerequisite	**Customer Defaults→Modeling→Feature Group→Prefer Sequential Timestamp Order** This option must be selected in order to see the **Allow Automatic Reordering of Feature Groups** check box in **Modeling Preferences**.
Menu	**Format→Group→Feature Group→Output** list **Preferences→Modeling→General** tab→**Allow Automatic Reordering of Feature Groups** check box

Curve analysis using preview curves

What is it?

You can now analyze curves for the following commands as you create or edit them using their dynamic preview curves:

- **Intersection Curve**
- **Offset Curve** (now newly supporting dynamic preview curves)
- **Project Curve**
- **Curve Length**

To see a curve analysis when preview curves are displayed, click an analysis option on the **Analyze Shape** toolbar.

Curve analysis for preview curves already exists for a number of curve commands, such as **Isoparametric Curve** and **Studio Spline**.

Why should I use it?

Analyze the shape of a curve you are creating or editing by dynamically displaying a curvature comb, curvature peak points, or curvature inflection points on the previewed curve.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Analyze Shape® Curve Analysis Display Drop-down** menu or **Comb Options**
Menu	**Analysis® Curve® Comb Options..**

Shape Studio

Fit Surface

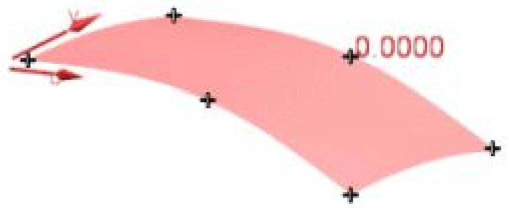
Use the **Fit Surface** command to fit surfaces to facet bodies, point sets or point groups as part of a reverse engineering workflow.

You can fit five types of surfaces.

Note In the following examples, the same point set was used to fit types. The maximum fitting error and the location of the error is displayed on-screen.

Fit Freeform

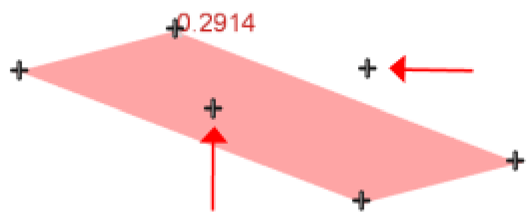
You can control degree and number of patches, their uniformity and fit direction. You can also control the boundary definition and smoothness of the fitted surface.



Fit Plane

You can specify a direction constraint where the plane normal will be parallel to the specified direction.

For any fitting operation except **Fit Freeform**, you can select the **Automatic Point Rejection** option and specify a rejection threshold distance so any points outside the threshold distance are not used for fitting a surface.

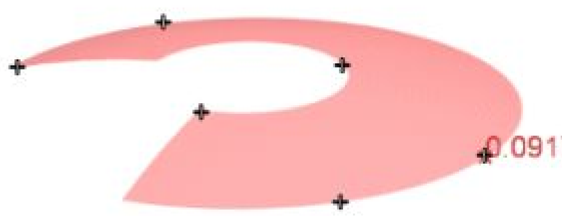


Automatic Point Rejection

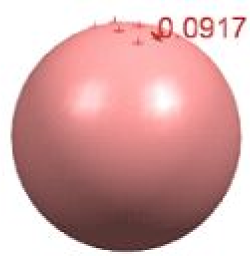
Fit Sphere

You can specify a radius value constraint.

You can create an open or closed spherical surface.



Closed

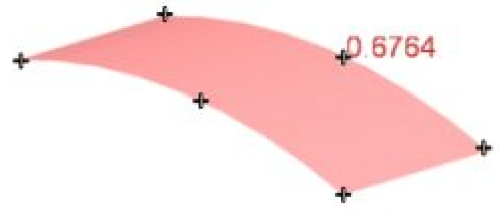


Closed

Fit Cylinder

You can specify a radius value constraint. If you apply a direction constraint, it will apply to the center line of the cylinder.

You can create an open or closed cylindrical surface.



Closed

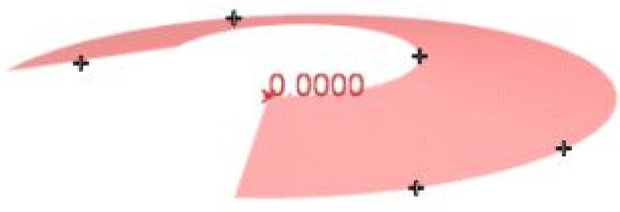
Closed

Fit Cone

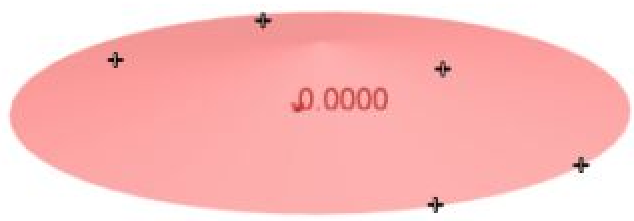
You can specify an angular constraint. When you have an angular constraint, the half angle is fixed to the value you specify.

If you apply a direction constraint, it will apply to the center line of the cone.

You can create an open or closed conical surface.



Closed



Closed

Fit Surface creates an editable feature. When editing the feature you can replace the target by selecting a different one.

Where do I find it?

Application	Modeling, Shape Studio
Toolbar	**Surface® Surface Drop-down® Fit Surface** 
Menu	**Insert® Surface® Fit Surface**

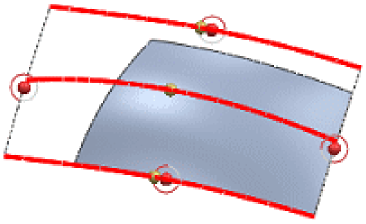
X-Form enhancements

What is it?

The workflow of the **X-Form** command has been enhanced with the following updates.

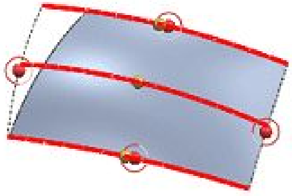
- The **Proportional** and **Falloff** advanced methods of previous releases have been combined into the new **Proportional** advanced method.
- You can now select poles and polylines from the graphics area when using the **Proportional** method.
- The **Keep Continuity** method has been relocated in the dialog box so you can now use it in conjunction with the core editing methods such as. **Move**, **Rotate**, **Scale** and **Planarize**.
- The **Keep Continuity** option now has on-screen continuity handles available at surface boundaries.
- An **Extraction Method** options list has been added to the **Settings** group.

Original

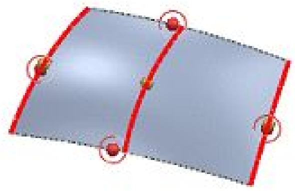


Extracts the face based on the original parametric, rectangular surface.

Minimum Bounded



Extracts the face based on a rectangular border enclosing the trimmed surface.

Fit to Boundary		Extracts the face by fitting a quadrilateral surface using the trimmed boundary of the trimmed surface.

- The **Feature Save Method** setting **Absolute** has been renamed to **Static**.
- You can now use **Edit Parameters** on an X-Form feature without rollback to facilitate 'design-in-context' workflows.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Edit Surface® Pole Editing Drop-down® X-Form** 
Menu	**Edit® Surface® X-Form**
Location in dialog box	**Method** group **Settings** group

Styled Blend and Silhouette Flange law option enhancements

What is it?

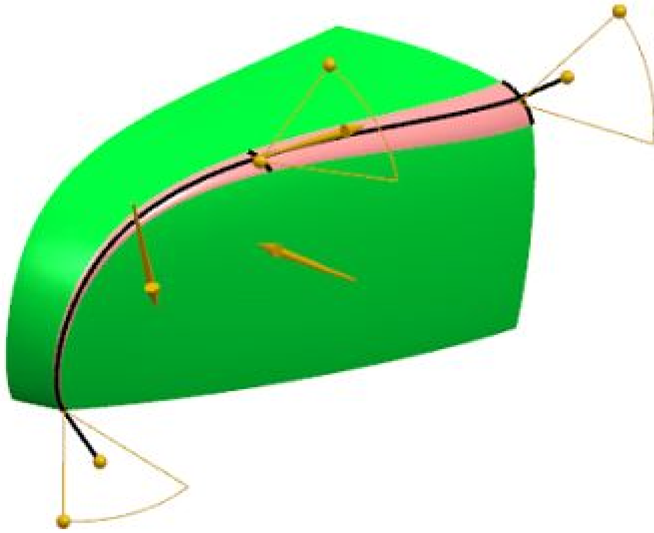
The law options of the **Styled Blend** and **Silhouette Flange** commands have been enhanced.

- The law types **Constant**, **Linear**, and **Cubic** have been added to both commands and are available for their respective control types.

Styled Blend
- o **Tube Radius 1**
- o **Tube Radius 2**
- o **Depth**
- o **Skew**
- o **Tangent Magnitude**

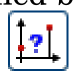
Silhouette Flange
- o **Radius**
- o **Length**
- o **Angle**

- The **Constant**, **Linear** and **Cubic** law types have dynamic manipulation handles available.



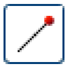
- There is a shortcut menu for parameter editing available on each respective law type handle.

- The dialog box options for the **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**  to denote there are no initial constraints.

Where do I find it?

Application	Modeling, Shape Studio
Commands	**Styled Blend** **Silhouette Flange**
Toolbars	**Feature® Styled Blend**  **Surface® Flange Surface Drop-down® Silhouette Flange** 
Menu	**Insert® Detail Feature® Styled Blend** **Insert® Flange Surface® Silhouette Flange**



Show End Points

What is it?

The **Show End Points** command lets you turn the display of the end points of curve features on and off.

You select curves, strings of curves, and curve features on-screen or from the **Part Navigator**.

You can specify preferences for the appearance of curve end points or change the customer defaults.

Filled Circle with **Curve Color** on.

Filled Circle with **Curve Color** off, **Color** = <any color>.

Open Circle with **Curve Color** on.

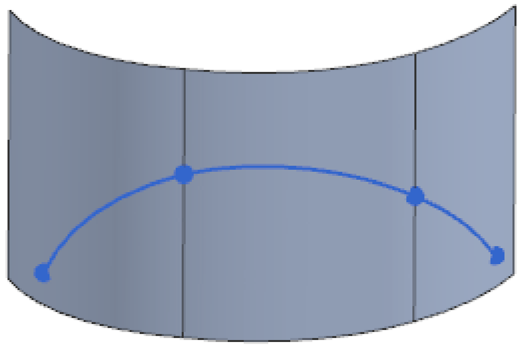
Plus Sign with **Curve Color** on.

Cross with **Curve Color** on.

Why should I use it?

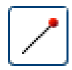
Use **Show End Points** when you want to graphically emphasize the location of curve end points of curve features.

It is also helpful to see how a string of curves is broken into segments.



Segment end points of projected curve

Where do I find it?

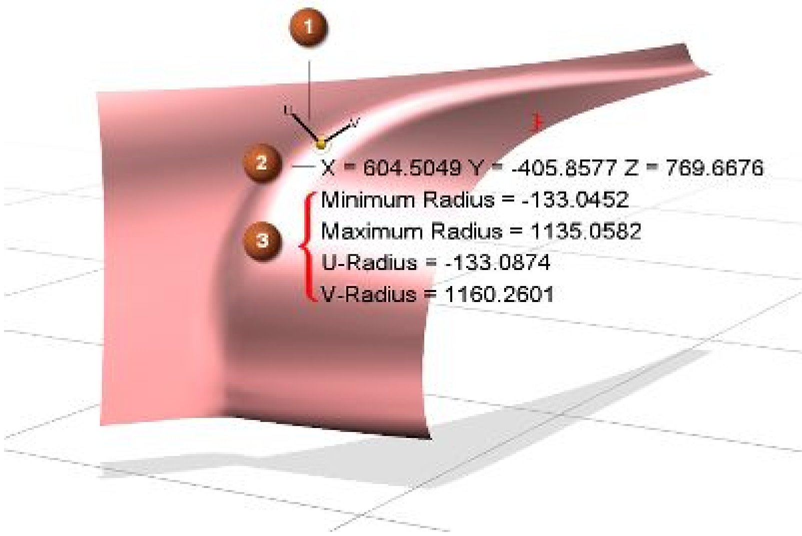
Application	Modeling, Shape Studio
Toolbar	**Analyze Shape® Show End Points** 
Menu	**Analysis® Shape® Show End Points**
Preferences	**Modeling® Analysis® Curve Display**
Customer Defaults	**Modeling® Freeform Modeling® Color and Font** tab® **Curve Display**

Local Radius Analysis

What is it?

Use the **Local Radius Analysis** command to select points on curves, edges, or faces and get back information about location, U/V parameters and radius of curvature.

- The new **Local Radius Analysis** command replaces the **Geometric Properties** command.
- **Local Radius Analysis** creates an analysis object that is associative with its parents so that as the parent is modified, the radius analysis information will update accordingly. The Local Radius analysis object is listed in the **Analysis** section of the **Part Navigator**.
- Radius analysis information is displayed on-screen and can also be displayed in an **Information** window.
- You can control what analysis information is displayed by selecting specific location, orientation and radius of curvature check boxes in the **Local Radius Analysis** dialog box.



1. **Orientation Display** = ☑ **U/V Tangent**
2. **Location Display** = ☑ **Coordinates**
3. **Radius of Curvature Display** = ☑ **Radius, Minimum/Maximum Radius, U/V Radius**

Where do I find it?

Application	Gateway, Modeling, Shape Studio
Menu	**Analysis® Local Radius Analysis**

Assemblies

Visual Reporting

Visual Reporting Enhancement for Assembly Navigator

What is it?

The interaction between the **Visual Reporting** tool and the **Assembly Navigator** has been enhanced in the following ways:

- You can now create a visual report directly from the **Assembly Navigator** by right-clicking a column that is a valid visual report property and choosing the **Report This Column** command. This generates a visual report based on that column.

- You can now configure an **Assembly Navigator** column with a part attribute. When you define a part attribute, a corresponding part attribute (work part) report is evaluated in the **HD3D Visual Report** dialog box.

The following **Assembly Navigator** columns have been added as properties to the **Visual Report Definition** dialog box:

- Arrangements
- Callout
- Count
- File Description
- Layer
- Out of Date
- Product Interfaces
- Shape

In Teamcenter mode, a new Reference-only column has been added to the **Assembly Navigator** under the **Component Property** source.

Why should I use it?

You can now see report results for color and tag components in the **Assembly Navigator** as soon as a report is generated. Previously, you had to first start the HD3D **Visual Reporting** tool or use quick report. You then had to return to the **Assembly Navigator** in order to see the component, its report group, and report property.

Where do I find it?

Application	Modeling and Assemblies
Assembly Navigator	Right-click a column ® **Report This Column**.

Teamcenter properties in HD3D Visual Reporting

What is it?

More property types are available for visual reports when you run NX with Teamcenter Integration, including:

- BOM line properties

- Item Master form properties
- Item Revision Master form properties

You can also create a visual report for a property on a secondary object related to an item or item revision by specifying relation or attribute types.

You can add the new property types to the list of available visual reports in the same way as other property types. Use the **Visual Report Definition** dialog box to add each property type that you want. You can then run a visual report on the property type using the HD3D **Visual Reporting** tool. For more information about visual reporting, see the Assemblies Help.

Why should I use it?

You can use the property types to run additional visual reports on your model and get more information.

Where do I find it?

Resource bar	**HD3D Tools**® **Visual Reporting**® **Define New Report** 
Location in dialog box	**Report Property** group® **Property** **Report Scope** group® **Terms**® **Property**

Visual Reporting enhancements

What is it?

The way that NX constructs the list of reports displayed in the Visual Reporting HD3D tool and in the **Visual Reporting** toolbar is changed.

- The number of recent reports included in the list is reduced from 20 to 10.
- The total list length is increased from 20 to unlimited. A scroll bar now appears when needed.

Why should I use it?

The performance of the report list is improved because:

- Construction of the report list is faster because fewer recent reports are pre-loaded.
- All reports that are provided with NX are now displayed in the list because of the increase in the total list length.

Where do I find it?

Resource bar	**HD3D Tools**® **Visual Reporting**® **Report** group® **Report Name**
Toolbar	**Visual Reporting**® **Report Name**

Teamcenter List of Values (LOV)

What is it?

The HD3D **Visual Reporting** tool has been enhanced so that when an LOV is attached to a Teamcenter property, you can now see the list and select one of its values when you define or run a visual report in managed-mode NX.

In Visual Reporting, NX displays the list of values defined for the following kinds of properties:

For these properties	Under these conditions
Teamcenter property	• A list of values is attached to the property in Teamcenter.
Part attribute	• The property is a part attribute defined in an NX attribute template • The data type is one of **Date**, **Double**, **Integer**, or **String**. • The attribute has constraint type **List** or **On**

Why should I use it?

Selecting the value for a Teamcenter property from an LOV instead of entering the value by typing has advantages such as the following:

- You know that the value you select is a legitimate value.
- You eliminate the risk of typographical errors.

Where do I find it?

Visual Report Definition dialog box

Resource bar	**HD3D Tools**® **Visual Reporting**® **Define New Report**
Location in dialog box	**Report Scope** group® **Terms**® **User Specified** **Results** group® **User Defined Groups**® **Value**

Visual Reporting tool

Resource bar	**HD3D Tools**® **Visual Reporting**
Location in dialog box	**Report Scope** group® **Terms**

Date visual reports

What is it?

When the report property is a date, in the **Visual Reporting Definition** dialog box, the **Group By** list for automatic or semi-automatic result grouping now contains time interval options. The options include:

Day
Week
Month
Year

Where do I find it?

Resource bar	**HD3D Tools**® **Visual Reporting**® **Report** group® **Define New Report** or **Edit Report** .
Location in dialog box	**Visual Reporting Definition** dialog box® **Results** group® **Groups** subgroup

Custom visual reports

What is it?

A customization guide is now available to help you learn how to modify or create your own Visual Reporting report formats, bitmaps, and information view templates.

The guide is in the Microsoft Word file *VisualReporting_Customization_Guide_v1.0.doc*.

Why should I use it?

Custom visual reports can help you quickly collect and display the particular information you want to review.

Where do I find it?

%UGII_BASE_DIR%\ugii\visual_reports\customization

Create Linked Mirror Part

What is it?

Use the **Create Linked Mirror Part** command to create an associative linked mirror part from a component in the **Assembly Navigator**. The source part can be either a piece part or an assembly component.

The linked mirror part has the same geometry, feature groups, object groups, attributes, layer structure, and optionally PMI, as the source part.

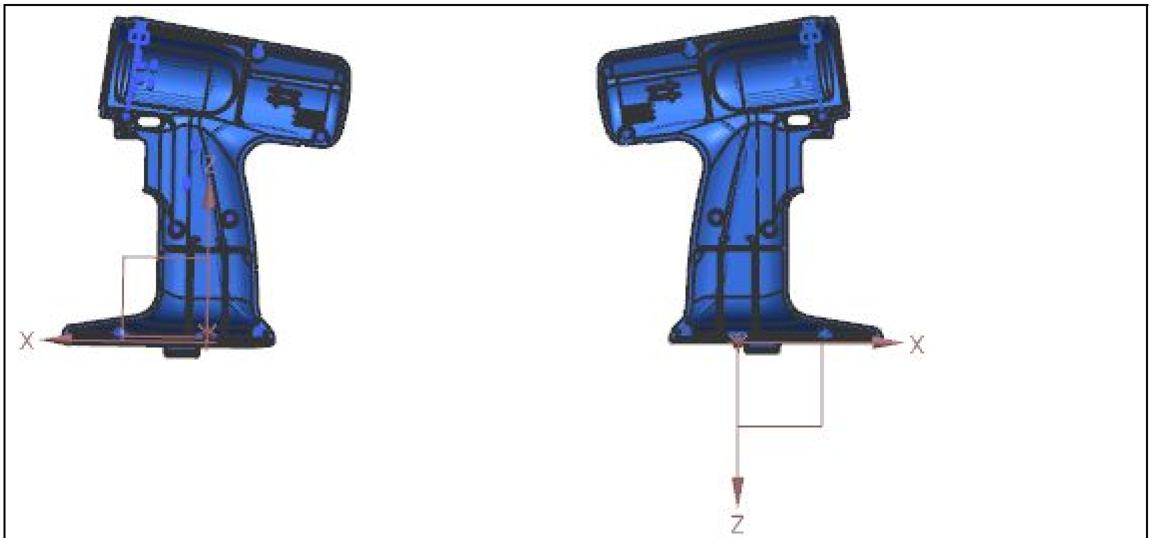
You can:

- Control which objects are mirrored by specifying the reference sets that contain the objects you want. If you select the **Entire Part** reference set, everything, including all reference sets, is mirrored.

- Control the type of linked mirror part that is created. There are two kinds of linked mirror parts: exact and non-exact. An exact mirror part is an exact replica of the source part. A non-exact mirror part can differ from the source part because of modifications made after its creation.

- Provide a template part as a seed part to be used for creating the linked mirror part.

- Modify the source part by adding, removing, modifying, or reorganizing geometry. These changes are applied to the mirror part when the mirror part is updated from the source part.



Source part Mirror part

If you modify an exact mirror part, all modifications are lost when the mirror part updates from the source part. Modifications made to a non-exact mirror part are not lost. Exact mirror parts are used most often.

When you create a non-exact mirror part, you can specify whether broken links are deleted or kept when the source part is modified.

- If you delete broken links, features that are dependent on the linked geometry may also be deleted.
- If you keep broken links, you have a chance to reparent dependent features. The broken links remain until you manually delete them.

Coordinate systems in NX are right-handed. If the part being mirrored includes a coordinate system, NX mirrors two axes and derives the third axis from the **CSYS Mirror Method** option. In the figure, **CSYS Mirror Method** is set to **Mirror X and Y, Derive Z**.

Why should I use it?

The **Create Linked Mirror Part** command is useful when you want to create an associative mirror part whose data (geometry, PMI, and user-attributes) and organization (object-groups, feature-groups, and reference sets) are associative to the source part.

Where do I find it?

Prerequisite	Right-click in the background of the **Assembly Navigator** and ensure that **WAVE Mode** is on.
Assembly Navigator	Right-click a component node® **Create Linked Mirror Part**

Constraint groups

Use the **New Constraint Group** command to create constraint groups in the **Assembly Navigator** or **Constraint Navigator** to organize the constraints in your assembly. A constraint group associates a set of assembly constraints, and is displayed as a single item in the navigator.

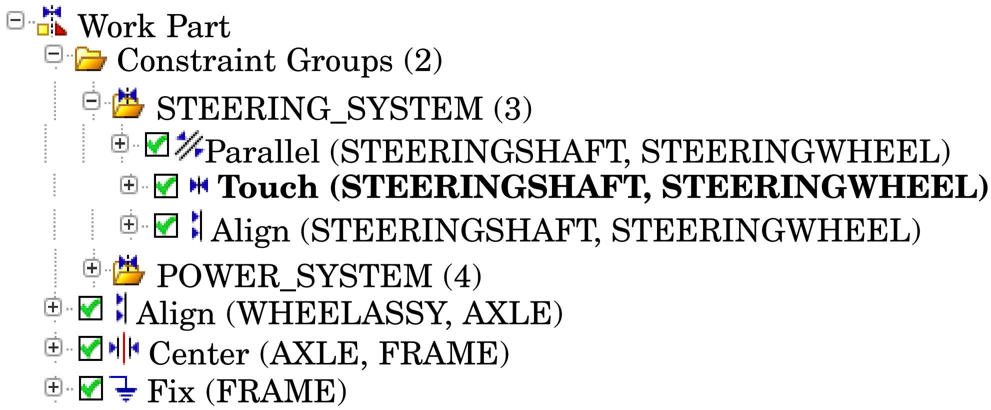
Constraint groups are useful in situations such as the following:

- You want to identify a set of constraints with a particular connection.
- You want to organize the constraints in your assembly.

To add constraints to a constraint group, you can select constraints or components. If you select components, you can add either the constraints between the selected components, or the constraints connected to the selected components.

The following figure shows two constraint groups: **steering_system** and **power_system**. The **Constraint Navigator** mode is **Group by Constraints**.

Constraint Navigator



A constraint can belong to more than one constraint group. In both navigators, each constraint is displayed under every constraint group to which it belongs. All constraints continue to be displayed under the **Constraints** node in the **Assembly Navigator** and the **Work Part** node in the **Constraint Navigator**.

You cannot drag a constraint directly to a constraint group in the navigators. You must use the **Constraint Group** dialog box options.

You can edit a constraint group by right-clicking its node and choosing a command from the shortcut menu. You can also edit a constraint group from the **Format® Group** menu, if you customize the menu to add the **Edit Constraint Group** option.

Where do I find it?

Application	Assemblies
Menu	**Format® Group® New Constraint Group**
Assembly Navigator	Right-click the **Constraints** node® **New Constraint Group**
Constraint Navigator	Right-click the **Work Part** node® **New Constraint Group**

Constraint name display

What is it?

You can use the new **Display Constraint Names** list to specify how constraint names are displayed in the **Assembly Navigator** or the **Constraint Navigator** when a constraint has a user-defined name. The constraint name can be displayed as system name only, user-defined name only, or a combination of both names.

For example, if you give the user-defined name **AXIAL** to an **Align (Nut, Bolt)** constraint, depending on your **Display Constraint Names** choice, the constraint name is displayed as one of the following.

Display Constraint Names option	Constraint name displayed in navigator
System and User	**Align (Nut, Bolt) “AXIAL”**
User and System	**“AXIAL” Align (Nut, Bolt)**
System Only	**Align (Nut, Bolt)**
User Replaces System	**“AXIAL”**

Constraints that do not have a user-defined name are shown with the system name, regardless of the **Display Constraint Names** setting.

The initial setting of the **Display Constraint Names** list is controlled by the **Constraint Name Display** customer default.

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

Why should I use it?

You can more easily find the constraints you want in the navigator.

Where do I find it?

Assembly Navigator	Right-click in the background® **Properties**
Constraint Navigator	Right-click in the background® **Properties**
Location in dialog box	**General** tab® **Display Constraint Names**

Position Independent Linked Object - check box

What is it?

The **Make Position Independent** option is now available in the following situations:

- When you create a link in context of an assembly.
- When you edit a link for **Region of Faces** features and **Mirror Body** features.

Previously, the **Make Position Independent** check box was available only on the edit version of the **WAVE Geometry Linker** for linked Composite Curves, Sketches, Faces, and Bodies.

Why should I use it?

If you know you want a position-independent linked object (PILO), you can save time by specifying this when you initially create a WAVE link. Previously, a WAVE link created in context of an assembly was always position-dependent, so you had to convert it to a PILO after creating it.

Where do I find it?

Application	**Modeling**
Menu	**Insert**® **Associative Copy**® **WAVE Geometry Linker**
Location in dialog box	**Settings** group® **Make Position Independent**

Expression formula locking and filtering enhancements

Expression formula locking enhancements

You can now lock and unlock one or more expression formulas in the **Expressions** dialog box using the new **Lock Formula** option. The expression values can still update while the formulas are locked.

Expressions with locked formulas are grayed out and display the lock icon .

Caution An overriding interpart expression can override the value of an expression, ignoring the Lock Formula.

Note To unlock a locked expression, right-click the expression and choose **Unlock Formula**.

Expression filtering enhancements

- You can now use the **Object Parameters** option to filter for the following types of expressions associated with features:
  - String feature expressions
  - Vector feature expressions
  - Point feature expressions
  - Boolean feature expressions
  - Suppression controlling expressions
- You can now use the **Filter by Feature Type** option to filter for feature expressions of a selected feature type in a model.

The **Filter by Type** category is now called **Filter by Expression Type**.

Where do I find it?

Application	Modeling and Assemblies
Menu	**Tools→Expression**
Location in dialog box	Locking or unlocking: Right-click an expression® **Lock Formula** or **Unlock Formula** Filtering: **Listed Expressions** group® Categories list

Create Multiple Interpart Expressions

What is it?

Use the new **Create Multiple Interpart Expressions** option to simultaneously link multiple expressions from a source part into the current work part.

You can still create a single interpart expression using the new **Create Single Interpart Expression** option. To edit interpart expressions you can use the new **Edit Multiple Interpart Expressions** option.

Naming rule options

You can choose from several new automated naming options when you use **Create Multiple Interpart Expressions**.

- **Add Prefix/Add Suffix** adds a prefix or suffix to the source expression name when the interpart expression is created.

- **Replace** lets you specify a name to find and replace with another name of your choosing.

- **Replace with Index** lets you replace the source expression name with a name of your choosing. As more interpart expressions are created an index is added to the name.

Where do I find it?

Application	Modeling and Assemblies
Menu	**Tools→Expression**
Location in dialog box	**Create Single Interpart Expression** **Create Multiple Interpart Expressions** **Edit Multiple Interpart Expressions**

Drafting

DraftingPlus enhancements

Drawing Booklets

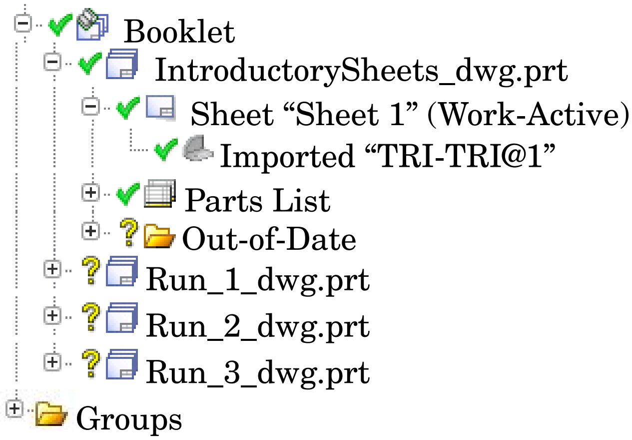
A drawing booklet is a structure of multiple files, each comprised of one or more drawing sheets, which together make up a complete set of drawings required to define the entire assembly or study. The files in the booklet are collectively created and managed as a single unit.

You can use booklets to break up very large assembly and detail drawings (with 125 or more drawing sheets) into more manageable drawings across multiple parts. This reduces the impact large drawings can have on system resources.

For example, a single booklet may consist of multiple detail drawings generated for an entire assembly, or it can be restricted to a selected set of components within the assembly. The booklet itself can be further compartmentalized into individual part containers, each containing a subset of sheets. As with components in general, a detail part can be included in more than one drawing booklet.

You can generate a booklet in both the native and managed environments. In managed mode, the booklet is represented as a Teamcenter object containing the Items and Revisions that hold the part containers which collectively represent the entire drawing. Booklets in managed mode appear in a Teamcenter Navigator folder.

Booklets in native mode are organized in the **Part Navigator** under a *Booklet* node. If the booklet has been compartmentalized, separate nodes for each container appear under the booklet node. Each drawing node lists the individual sheets in it. View nodes appear under each sheet node. If there is a parts list on a sheet, a node for it is also created.



Creating and identifying booklets

Drawing booklets are created with the **Create Drawing Booklet** wizard. The booklet name helps identify the collection of parts that together make up the entire drawing. In the managed mode, the booklet name is used for relating the Item and Revisions together and for identifying and locating these items.

Numbering sheets

You can specify the first sheet number for booklets and also edit the sheet number of a drawing sheet. When the sheet number is changed, the booklet automatically rearranges the order of the sheets in the work part.

Plotting

The booklet provides utilities that let you print or plot the collection of sheets in it. You can select one sheet, all sheets, or a set of sheets to print.

Editing drawing booklets

You can use the **Edit Drawing Booklet** command to edit an existing booklet and add drawings based on a set of user inputs and predefined rules.

Where do I find it?

Create new booklets:

Application	All applications except Drafting
Menu	**Tools→Drawing Automation→Create Drawing Booklet**

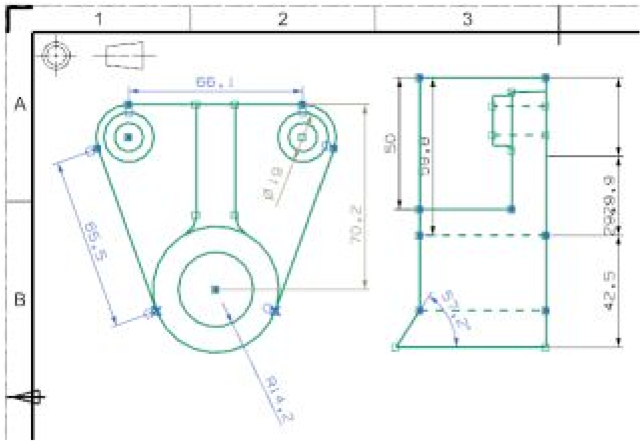
Edit existing booklets:

Application	All applications except Drafting
Menu	**Tools→Drawing Automation→Edit Drawing Booklet**

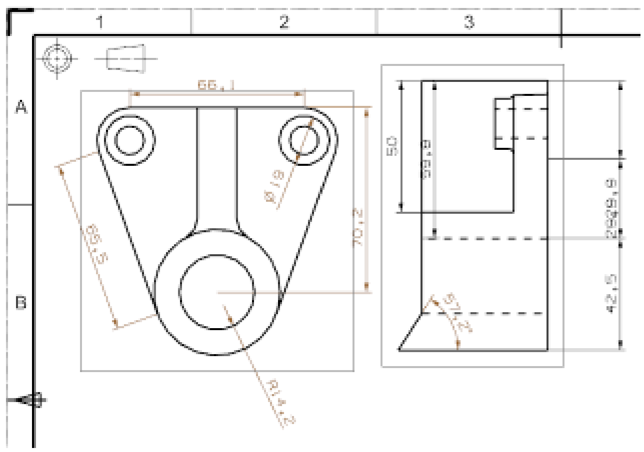
Move to New View

What is it?

Use the **Move to New View** command to move selected sheet objects to a new **Drawing** view.



Initial sketch on the sheet



Selected sketch objects placed in two new views

In the **Move to New View** dialog box, you specify:

- A set of objects to move. If you select annotations that reference hidden objects, these objects are included in the selection.

- The view coordinate system, scale, orientation, and style.

Objects maintain their apparent size and position. As a result, dimension values change whenever view scale is not 1:1.

If your selection contains part of a sketch, the sketch is split and the selected part is placed in a new view sketch. Sketch constraints that cannot be preserved when splitting are removed.

All annotations and dimensions remain view dependent on the sheet.

Why should I use it?

This command helps you reorganize the content of drawings into **Drawing** views to leverage a number of view-centric design workflows such as using the **Project to View** and **Copy to 3D** commands to create 3D models from 2D drawings.

Where do I find it?

Application	Drafting
Menu	**Edit**® **View**® **Move to New View**
Graphics window	Right-click sketch curves® **Move to New View**
Part Navigator	Right-click the sheet sketch node® **Move to New View**

Smash View

What is it?

Use the **Smash View** command to move view dependent objects from a view onto the sheet. You can move objects from the following views:

- **Drawing** views
- **Detail** views created from **Drawing** views

If you smash a **Drawing** view that has an associated **Detail** views, the **Detail** views become independent.

Note The original view is removed from the drawing.

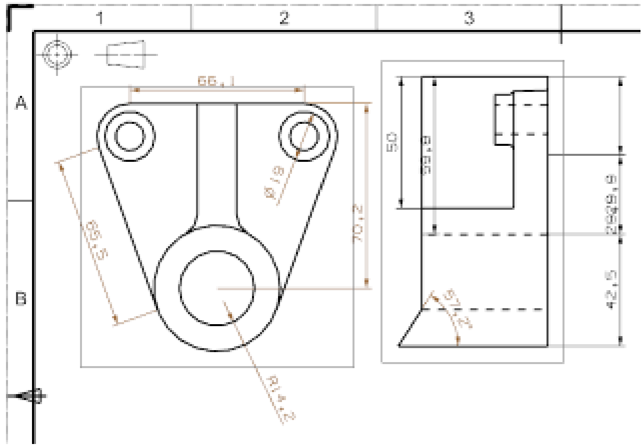
Objects maintain their apparent size and position. As a result, dimension values change whenever view scale is not 1:1.

Hidden objects are also transferred onto the sheet and view clipping is removed.

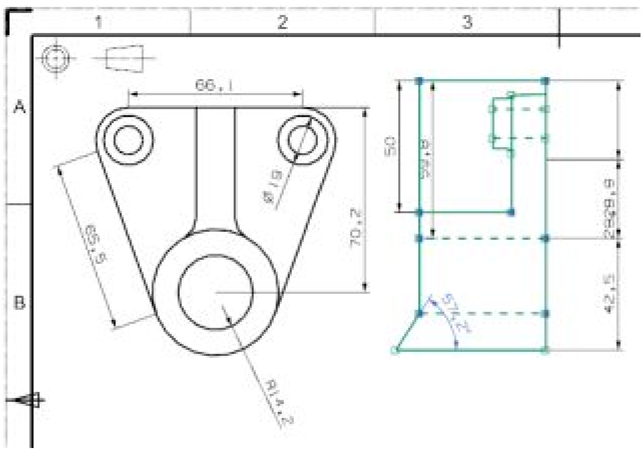
The following view dependent objects are deleted:

- Retained annotations
- Tabular notes

Exact annotation orientation cannot be maintained when the original view is rotated, due to constraints imposed on sheet annotations.



Drawing views to smash



View dependent objects on the sheet

Why should I use it?

This command helps you reorganize the content of layout by moving content contained in a view to a sheet. This provides a more flexible 2D design environment.

Where do I find it?

Application	Drafting
Menu	**Edit® View® Smash View**
Graphics window	Right-click sketch curves® **Smash View**
Part Navigator	Right-click the sheet sketch node® **Smash View**

Importing I-deas curves into sketches

What is it?

Use the new **Import Sheet Curves to Sheet Sketch** and **Import View Curves to View Sketch** check boxes to import I-deas curve geometries to related sheet or view sketches.

Why should I use it?

Migrating I-deas curve geometries directly to NX sketches during the import phase simplifies your workflow and lightens your work load.

Where do I find it?

Application	Drafting
Menu	**File**® **Import**® **I-deas ASC/DWG**
Location in dialog box	**Options** group® **Import Sheet Curves to Sheet Sketch** or **Import View Curves to View Sketch**

Custom symbol smash behavior

What is it?

Use the new **Smash to Sketch** check box to generate sketch curves when you smash a custom symbol.

This option works whether or not the instantiated custom symbol was originally created with sketch geometry.

Why should I use it?

Smash to Sketch ensures that the resultant geometry created from a smashed custom symbol is sketch geometry that can be used in a 2D design process.

Where do I find it?

Smash to Sketch drafting preference

Application	Drafting
Menu	**Preferences**® **Drafting**
Location in dialog box	**Custom Symbols** tab® **Smash to Sketch**

Smash to Sketch customer default

Menu	**File**® **Utilities**® **Customer Default**

Location in dialog box	**Drafting® Custom symbols® All** tab® **Smash to Sketch**

Lightweight drafting views

What is it?

The new **View Configuration** options let you automatically create exact or lightweight drafting views. Except for **Break-out Section** views and those containing view breaks, lightweight drafting views, which contain JT faceted representations of model geometry, can be created for all drafting views.

When the master model part is loaded into your session, you can annotate a lightweight view as you would any other drafting view. All dimensions created in the lightweight views are true dimensions. Plotting is also supported for lightweight views.

Exact drafting views support all view style options. **Lightweight** drafting views support view style options with the following exceptions. You cannot:

- Apply UV grids to the view.
- Edit the tolerance of the view.
- Apply **Referenced Edges**, **Self Hidden**, or **Small Features** options to hidden edges in the view.
- Apply a perspective to the view.
- Apply shading to the view.
- Apply end gaps to smooth edges in the view.
- Display threads in the view.
- Display virtual intersections in the view.

Normal and faceted representation views created in earlier versions of NX are now considered **Legacy** views. You cannot convert **Legacy** views to **Exact** or **Lightweight** views.

Why should I use it?

Lightweight views are used to improve memory usage and system performance when constructing drawings of large assemblies.

Where do I find it?

Set view type

Application	Drafting
Toolbar	**Drawing**® **View Preferences** 
Menu	**Preferences**® **View**
Location in dialog box	**General** tab® **View Configuration** group® **Representation**

Create drafting views with non-associative extracted edges

Application	Drafting
Toolbar	**Drawing**® **View Preferences** 
Menu	**Preferences**® **View**
Location in dialog box	**General** tab® **View Configuration** group® **Snapshot**

Enable creation of **Legacy** exact and lightweight views

Application	Drafting
Prerequisite	The **Legacy** option must be set in the **File**→**Utilities**→**Customer Defaults** dialog box, on the **Drawing**→**View**→**General** tab
Toolbar	**Drawing**® **View Preferences** 
Menu	**Preferences**® **View**
Location in dialog box	**General** tab® **View Configuration** group® **Legacy**

Set the default quality of the new lightweight drafting views

Menu	**File**→**Utilities**→**Customer Defaults**
Location in dialog box	**Drawing**→**View**→**General** tab® **Lightweight View Quality**

Drafting user interface enhancements

What is it?

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

- Unnecessary Modeling toolbars and commands are removed when a drawing sheet is visible. If applicable, these toolbars and commands become available when the drawing sheet is suppressed.
- The **Drafting** toolbar is only available in full screen mode.
- The **Display Sheet** command is removed from all toolbars. If the command is needed, you can add it to any toolbar using the **Customize** dialog box.
- Selection options that are not required in the drafting environment, such as **Create Interpart Link**, are no longer displayed.
- Unnecessary **Part Navigator** nodes, such as **Model Views**, are suppressed while in the Drafting environment with a sheet displayed.
- Modeling view names have been standardized to **Top**, **Front**, **Right**, **Back**, **Bottom**, **Left**, **Trimetric**, and **Isometric**. Drafting view names are derived from these standardized names.
- The **Identification Symbol** command is renamed **Balloon**.
- The **Show** and **Hide** command now supports all drafting annotations.
- The **Inherit** option is added to the **Feature Control Frame**, **Datum Feature Symbol**, **Datum Target**, and **Surface Finish Symbol** dialog boxes.

Where do I find it?

Balloon command

Application	Drafting
Toolbar	**Annotation® Identification Symbol**
Menu	**Insert® Annotation® Identification Symbol**

View Creation Wizard enhancements

What is it?

Several improvements have been added to the **View Creation Wizard** to enhance its overall performance and usability, especially in the realm of large assembly drawings.

Part step

The **Part** step displays a message window advising you to select a master model part to add to your drawing and then directs you to the **Part Name** dialog box so you can select one. This occurs when your active part contains no assembly components and is intended to encourage you to use master model architecture when creating your drawing. You can either select an existing part from the **Part Name** dialog or click **Cancel** to return to the **Part** step.

Large Assembly step

The **Large Assembly** step is a new step dynamically added to the **Task Navigator** when the **Show Large Assembly Option if Components Exceed** option is set, and the number of components in the master assembly part exceeds the value given for the option. The display settings in the **Large Assembly** page take precedence over those settings in the **Options** page and are designed to maximize system performance while placing large assembly views.

You can:

- Use the **Automatically Optimize Settings** option to automatically select the optimum **View Configuration** and **Custom View Settings** options.
- Control the display quality of exact or lightweight views.
- Control view previews using the **Set View Preview to Border** option.
- Use the **Custom View Settings** option to manually override traceline, smooth edge, hidden line, thread, and virtual intersection displays.

Options step

When the **Large Assembly** step is displayed, its options supersede those in the **Options** page, and the following options are removed:

- **Process Hidden Lines**.
- Hidden line **Color**, **Font**, **Width** drop-down lists.
- **Show Centerlines**.
- **Preview Style** drop-down list.

When **Automatically Optimize Settings** is selected in the **Large Assembly** page, the **View Style** settings on the **Options** page are disabled.

Orientation step

- To improve performance, view boundaries are only calculated one time as the parent view is selected.

- When you use the **Customized View** option to define a non-standard view orientation, a **Customized View** item is added to the **Model Views** list. This item remains in the **Model Views** list until you close the **View Creation Wizard**.

Layout step

- With the **Manual** placement option, a view scale appears on the **Status Line** as you position your views on the drawing sheet. The scale remains on the **Status Line** and continually updates until you complete your view placement.

- When **Ignore Title Blocks** is selected, the title block area is ignored when you place a view. The view and any inherited PMI can intrude into the area of the title block. By default this option is not selected.

- The view button arrangement in the **Layout** page is more intuitive and more accurately reflects the corresponding view layout on the drawing sheet.

- View boundaries are only recalculated one time when changing **Placement** from **Manual** to **Automatic**, or when changing margin values.

Why should I use it?

The performance improvements added to the **View Creation Wizard** give a greater degree of control in managing large assembly views and utilize the lightweight view capabilities introduced in NX 8.5.

Where do I find it?

Application	Drafting
Toolbar	**Drafting® View Creation Wizard** 
Menu	**Insert® View® View Creation Wizard**

Activate the **Large Assembly** step:

Application	Drafting
Prerequisite	The **Show Large Assembly Option if Components Exceed** option must be set in the **File→Utilities→Customer Defaults** dialog box, on the **Drawing→General→View** tab
Menu	**Preferences® Drafting**
Location in dialog box	**View** tab® **View Creation Wizard** group® **Show Large Assembly Option if Components Exceed**

Associative view alignment

What is it?

Use the new **Associative** check box to permanently align a view with other views when placing a new view or editing existing views. The associative alignment enforces the alignment even when views change or move.

You can add associative alignments to the following views provided that the placement method is not inferred:

- Base views
- Drawing views
- Projected views
- Detail views

You can add new associative alignments to existing views, including Section views, or edit existing associative alignments using the new **Alignment** command. The **Section View** dialog box does not have the **Associative** check box.

The **Associative Alignment** customer default sets the default value for the **Associative** check box.

Why should I use it?

Use associative alignments to make sure that views remain properly aligned on the drawing sheet after model modifications or view boundary changes.

Where do I find it?

Associative alignment in view commands

Application	Drafting
Prerequisite	The placement method must not be set to **Infer**.
Toolbar	**Drawing® Base View** or **Drawing View** or **Projected View** or **Detail View** **Drawing® Add View Drop-down** list® **Base View** or **Drawing View** or **Projected View** or **Detail View**
Menu	**Insert® View® Base** or **Drawing** or **Projected** or **Detail**
Shortcut Menu	Right–click a drawing sheet border® **Add Base View** or **Add Drawing View** or **Add Projected View** or **Add Detail View**
Part Navigator	Right–click a sheet node® **Add Base View** or **Add Drawing View** or **Add Projected View** or **Add Detail View**
Location in dialog box	**Placement** subgroup® **Associative**

Alignment command

Application	Drafting
Menu	**Edit® View® Alignment**
Graphics window	Right-click view border® **Alignment**
Part Navigator	Right-click the view node® **Alignment**

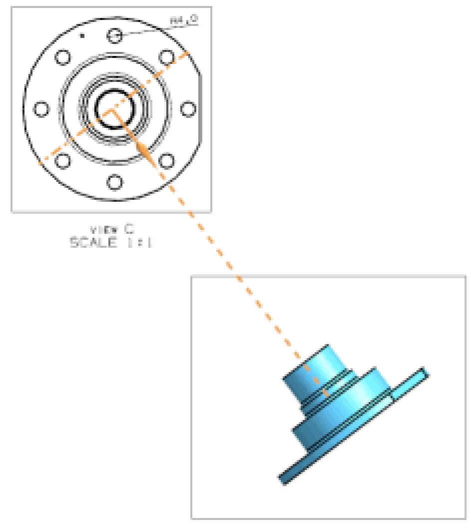
Associative Alignment customer default

Menu	**File® Utilities® Customer Default**
Location in dialog box	**Drafting® General® View** tab® **Associative Alignment**

Hinge view alignment method

What is it?

Use the new **Hinge** alignment method to easily place a Projected view when the parent view is on the same sheet.



The hinge alignment method is also available from the new **Alignment** command.

Where do I find it?

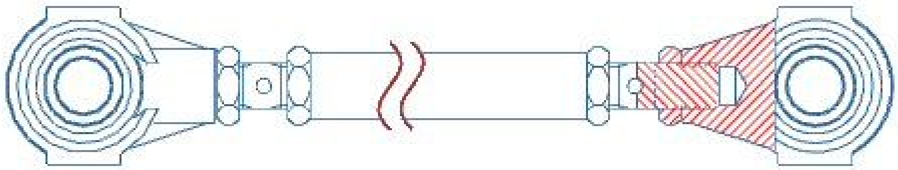
Application	Drafting
Toolbar	**Drawing® Projected View** **Drawing® Add View Drop-down** list® **Projected View**
Menu	**Insert® View® Projected**
Shortcut Menu	Right–click a drawing sheet border® **Add Projected View**
Part Navigator	Right–click a sheet node® **Add Projected View**
Location in dialog box	**View Origin** group ® under **Placement** ® **Method** list ® **Hinge**

Break out section views with view breaks

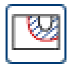
What is it?

You can now create a break out section view from a view that contains view breaks.

Note Sketch curves cannot be added to views that contain view breaks. You must either create the break out boundary curves before you add the view breaks, or suppress the view breaks and then create the curves.



Where do I find it?

Application	Drafting
Toolbar	**Drawing**® **Add View Drop-down** list® **Break-out Section View** 
Menu	**Insert**® **View**® **Section**® **Break-out**

Update enhancements for associative custom symbols

What is it?

You can now update unlocked, nested custom symbols in both Drafting and PMI.

You can set the new **Automatic** Drafting preference when creating an instance of a master custom symbol so that the symbol instance automatically updates when the master custom symbol changes.

Note PMI custom symbol instances are not supported for automatic update. You must use the **Out-of-date** node on the **Part Navigator** to update PMI custom symbol instances.

NX automatically updates unlocked custom symbols when:

- You start the Drafting application.
- The sheet that contains the custom symbol instances is updated.
- You update the view. This is true for symbol instances that are placed in an annotation view or are associated to a view.
- You save edits to a master custom symbol.
- You edit a nested custom symbol instance that contains other unlocked custom symbol instances.

Why should I use it?

Set your custom symbol update preference to **Automatic** if you want your unlocked custom symbol instances to always reflect the current master custom symbol definition.

Where do I find it?

Automatic update preference

Application	Drafting
Menu	**Preferences**® **Drafting**
Location in dialog box	**Custom Symbols** tab® **Automatic**

Automatic update customer default

Menu	**File**® **Utiltiies**® **Customer Default**
Location in dialog box	**Drafting**® **Custom symbols**® **All** tab® **Automatic Update**

Replace Custom Symbol

Use the **Replace Custom Symbol** command to:

- Change the associative link to a master custom symbol for one or more unlocked custom symbol instances. The custom symbol instances can include nested custom symbol instances.
- Repair a link to a master custom symbol.
- Create an associative link to a different master custom symbol.
- Create an associative link from a legacy custom symbol to an existing master custom symbol.

You cannot replace the associativity of a smashed custom symbol. You also cannot replace the associativity of a custom symbol embedded in a text string. However, you can replace all unlocked instances of selected symbols on the current drawing sheet by selecting the **Replace all Instances** option in the **Replace** dialog box. This includes instances embedded in a text string.

Where do I find it?

Application	Drafting
Toolbar	**Symbol® Replace Custom Symbol**
Menu	**Edit® Symbol® Replace Custom Symbol**
Graphics window	Right-click custom symbol instance® **Replace**

Custom symbol smash behavior

What is it?

Use the new **Smash to Sketch** check box to generate sketch curves when you smash a custom symbol.

This option works whether or not the instantiated custom symbol was originally created with sketch geometry.

Why should I use it?

Smash to Sketch ensures that the resultant geometry created from a smashed custom symbol is sketch geometry that can be used in a 2D design process.

Where do I find it?

Smash to Sketch drafting preference

Application	Drafting
Menu	**Preferences® Drafting**
Location in dialog box	**Custom Symbols** tab® **Smash to Sketch**

Smash to Sketch customer default

Menu	**File® Utilities® Customer Default**
Location in dialog box	**Drafting® Custom symbols® All** tab® **Smash to Sketch**

Width options for lines and symbols

What is it?

Additional line width options are available to control the appearance of lines and symbols in NX. All commands and preferences that contain the **Width** or **Line Width** option to define the line or symbol now provide these options.

List of available width options for lines and symbols

The original options of **Thin**, **Normal**, and **Thick** are represented by the **0.13 mm**, **0.18 mm**, and **0.25 mm** options, respectively.

Why should I use it?

Per industry or company standards, more than three line widths may be required when annotating a model or drawing. Varying line widths let you emphasize or de-emphasize geometry and symbols as needed.

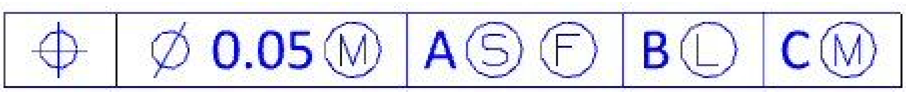
Where do I find it?

The width options are available in every dialog box in Drafting and PMI that contain a **Width** or **Line Width** option to define the size of the line or symbol.

Standard font support for symbols

What is it?

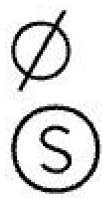
When the Drafting or PMI text type is set to a standard font type, symbols embedded in a note or other text are also presented in a font style similar to the text font. If a standard font representation does not exist for a symbol, the symbol is displayed as single-stroked lines. NX font types, such as **blockfont**, still display the symbols as single-stroked lines.



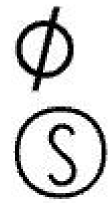
NX 8.0 and earlier

NX 8.5

Symbol font files, which are provided with NX, map the display of a symbol to standard font types. Symbol font files are provided for all of the supported drafting standards, and are set by a **Symbol Font File** annotation preference and customer default.



Example of ANSI symbols



Example of ISO symbols

Where do I find it?

Symbol Font File preference

Application	Drafting
Toolbar	**Annotation**® **Annotation Preferences** 
Menu	**Preferences**® **Annotation**
Location in dialog box	**Symbols** tab® **Symbol Font File**

Symbol Font File customer default

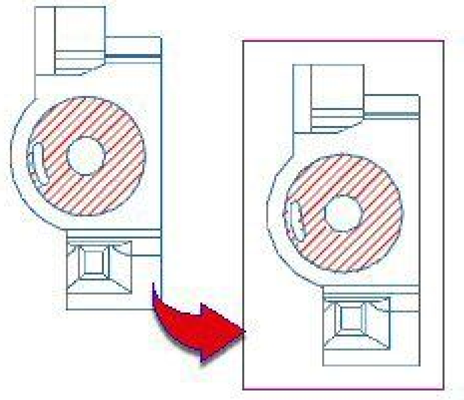
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Drafting**® **General**® **Standard** tab® click **Customize Standard**® **Annotation**® **Symbols** tab® **Symbol Font File** Note that you must start a new NX session before the change is available.

Crosshatch and Area Fill enhancements

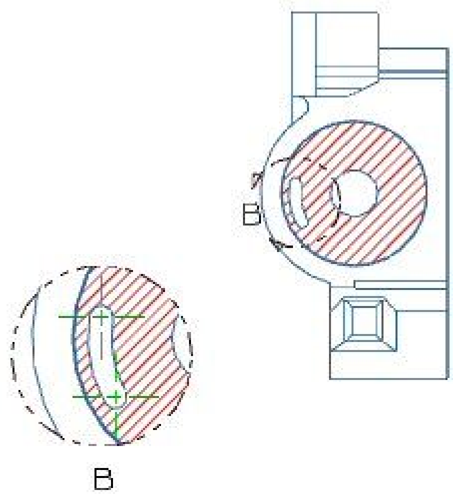
What is it?

The following enhancements have been made to the **Crosshatch** and **Area Fill** commands.

- When you copy and paste views created without extracted edges, but contain crosshatch or area fill patterns, the patterns also appear in the copied view.

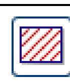
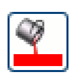


- When you create a detail view from a view created without extracted edges, but contains a crosshatch or area fill pattern, a portion of the pattern also appears in the detail view.



- All annotation types are now supported when you select the **Automatically Exclude Annotation** option.

Where do I find it?

Application	Drafting
Toolbar	**Annotation® Crosshatch**  **Annotation® Area Fill** 
Menu	**Insert® Annotation® Crosshatch** **Insert® Annotation® Area Fill**

Parts list and tabular note enhancements

What is it?

The following enhancements are made to the parts list and tabular note functionality.

- Improved highlight and selection interaction for cells, columns, and rows.
- The ability to identify the type of an angular value as radians or degrees.
- Better control over the font, width, and color of cell borders.
- The ability to apply the cell style setting from one cell to another cell using the **Inherit** command.
- The ability to navigate to a parts list section from the **Part Navigator**.
- The ability to interactively edit the slant angle of text in existing cells.
- General performance improvements for identifying attachment points for auto balloons, automatically sizing text in a cell, and updating parts lists.

Why should I use it?

These enhancements improve your interaction with, and optimize how you create and use parts lists and tabular notes.

Where do I find it?

Degrees or radian option for angular values

Application	Drafting and PMI
Prerequisite	Cell format must be set to **Degrees/Minutes/Seconds**
Toolbar	*Drafting only* **Annotation® Annotation Preferences**
Menu	**Preferences® Annotation**
Location in dialog box	**Cell** tab® **Type**

Slant angle option for individual cells

Application	Drafting and PMI
Toolbar	*Drafting only* **Annotation® Annotation Preferences** 
Menu	**Preferences® Annotation**
Location in dialog box	**Cell** tab® **Slant Angle**

Navigate to parts list section

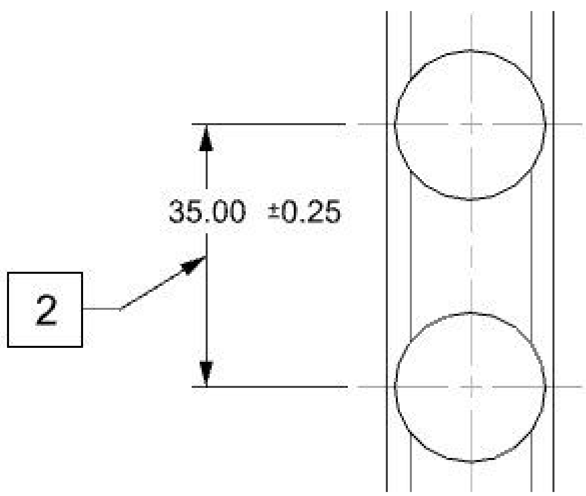
Application	Drafting
Part Navigator	Expand **Parts List** node® right-click **Parts List Section**® **Navigate to Section**

Leader line enhancements

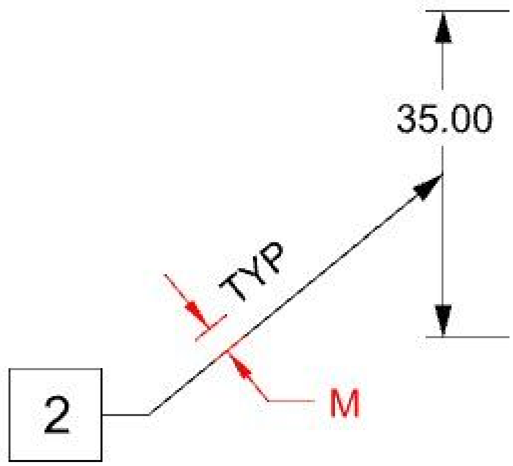
What is it?

Leader lines for Drafting and PMI annotations are improved with the following enhancements:

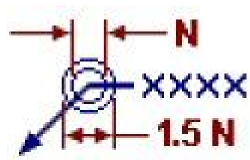
- You can associatively attach an annotation with **Plain** or **All Around** leaders to the midpoint or to a selected point of a dimension line.



- A new spacing option, **M**, lets you control the distance between a leader line and an annotation placed on the leader line using the **Flag** leader type. The spacing is given as a factor of the font size of the annotation you are placing.



- You can control the size of the **All Around** (single circle) and **All Over** (double circle) symbols on a leader line using the new **N** size option. The size is given in inches or millimeters, depending on the unit type of your part. The initial size is equal to the **Character Size** on the **Line/Arrow** tab of the **Annotation Preference** dialog box.



Where do I find it?

Set the **M** and **N** preference or edit the style of an existing leader

Application	Drafting and PMI
Toolbar	*(Drafting only)* **Annotation® Annotation Preferences**
Menu	**Preferences® Annotation**
Graphics window	Right-click leader line® Style
Location in dialog box	**Line/Arrow** tab® **M** **Line/Arrow** tab® **N**

M and **N** customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Drafting® General® Standard** tab® click **Customize Standard**. On the **Annotation® Line/Arrow** tab® **M** and **N**

Enhancements for view labels and section lines

What is it?

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

- Once all the available view letters have been used, you can add numbers instead of additional letters to a view label.

- You can display secondary view label annotation as subscript or inline text.

- You can display a rotation symbol and the rotation angle in the view label of a rotated section view.

  New angular precision options and angular format options can be applied to a view, and the values are also reflected in the view label.

- You can now assign one of eight different font styles to **ESKD Standard** type section lines.

Why should I use it?

Use these enhancements to create standards–compliant section labels and section lines.

Where do I find it?

Set the secondary index annotation, location, and size

Application	Drafting
Toolbar	**Drawing® View Label Preferences**
Menu	**Preferences® View Label**
Location in dialog box	**Settings** group® **Secondary Indexing** **Settings** group® **Secondary Indexing Alignment** **Settings** group® **Subscript Size Factor**

Customer Default for secondary index annotation, location, and size

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	On the **Drafting® General® Standard** tab, set **Drafting Standard** to **ESKD** then click **Customize Standard**. On the **View Label® Detail** tab, set the **Secondary Indexing**, **Secondary Indexing Alignment**, and **Subscript Size Factor** options. Note that you must start a new NX session before the change is available.

Display the rotation symbol and angle in the section label

Application	Drafting
Toolbar	**Drawing® View Label Preferences**
Menu	**Preferences® View Label**
Location in dialog box	**View Label** group® **Rotation Symbol** **View Label** group® **Rotation Angle**

Set angular precision and format options for a view

Application	Drafting
Toolbar	**Drawing**® **View Preferences**
Menu	**Preferences**® **View**
Graphics window	Right-click view border® **Style**
Location in dialog box	**General** tab® **Angle Precision** **General** tab® Nominal Angle Format **General** tab® **Leading Zeros** **General** tab® **Trailing Zeros**

Set the font style for the section line

Application	Drafting
Toolbar	**Annotation**® **Section Line Preferences**
Menu	**Preferences**® **Section Line**
Graphics window	Right-click the section line® **Style**
Location in dialog box	**Settings** group® **Standard**® **ESKD Standard** **Settings** group® **Font**

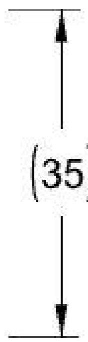
General annotation enhancements

What is it?

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

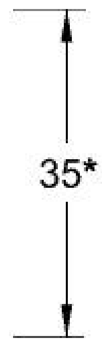
- Improvements to the selection of intersections and tangent points provide more robust and consistent results for dimension intersections and tangent point associativity.

- New preferences and customer defaults let you specify how to denote a Reference Only dimension.
  - o New **Reference Dimension** preferences let you specify a suffix or a prefix character instead of parentheses to denote a reference dimension. The default is to use parentheses to indicate a reference dimension.



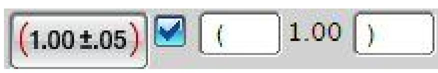
Default

Prefix only

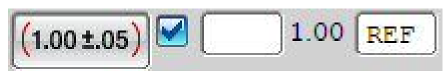


Suffix only

  - o Two new customer defaults, **Use 1.5 X Height Parentheses** and **Use Suffix/Prefix Characters**, let you specify the default preference for your NX session.



Reference Dimension preferences with parentheses default



Reference Dimension preferences with characters default

Where do I find it?

Reference Dimension preferences

Application	Drafting and PMI
Toolbar	*(Drafting only)* **Annotation**® **Annotation Preferences**
Menu	**Preferences**® **Annotation**
Location in dialog box	**Dimension** tab® **Precision and Tolerance** group® **Reference Dimension**

Reference dimension customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Drafting® General® Standard** tab® click **Customize Standard**. On the **Annotation® Dimensions** tab, select **Use 1.5 X Height Parentheses** or **Use Suffix/Prefix Characters**. Note that you must start a new NX session before the change is available.

Hole table enhancements

What is it?

You can now create a hole table for multiple bodies in one part. Holes from all the solids in the part will be added to the same hole table. All holes in the same solid will be indexed first, before holes in another solid are indexed. The first solid indexed is based on the selected ordinate origin point.

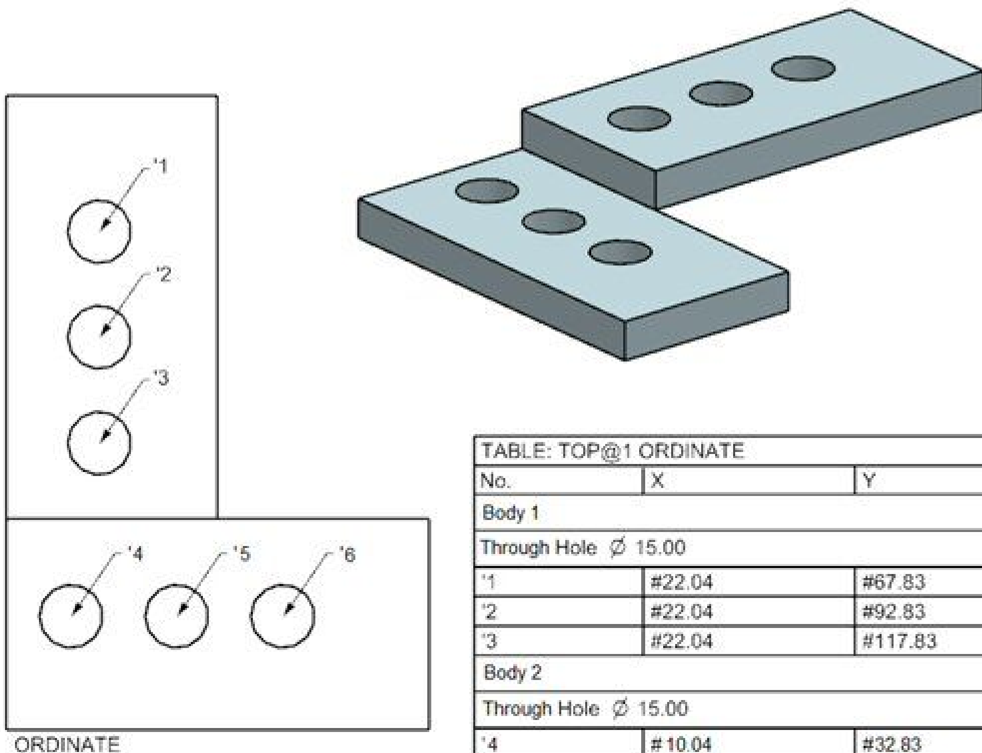
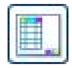


TABLE: TOP@1 ORDINATE		
No.	X	Y
Body 1		
Through Hole ∅ 15.00		
'1	#22.04	#67.83
'2	#22.04	#92.83
'3	#22.04	#117.83
Body 2		
Through Hole ∅ 15.00		
'4	# 10.04	#32.83
'5	# 35.04	#32.83
'6	#60.04	#32.83

Where do I find it?

Application	Drafting
Toolbar	**Table® Hole Table** 
Menu	**Insert® Table® Hole Table**

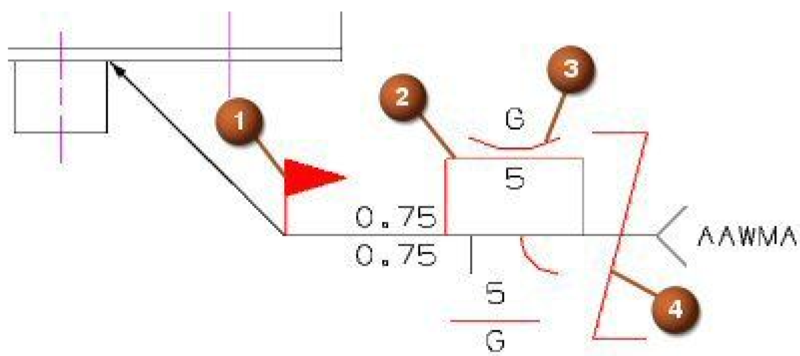
Weld symbol enhancements

What is it?

- A new **Weld Standard** option, available in the **Annotation Style** and **Annotation Preferences** dialog boxes, lets you interactively set the weld standard. This lets you access different standards-driven weld symbols without having to reset the entire drafting standard in your NX session.

- A new **Weld Symbol Size Factor** option lets you control the size of the symbols in a weld symbol. The size is given as a factor of the character size of the symbol, where 1.0 means the symbol is equal to the current character size.

  The size factor is applied to the field symbol (1), the weld symbol (2), the contour symbol (3), and the staggered annotation symbol (4) elements above and below the reference line of the weld symbol.



Where do I find it?

Application	Drafting and PMI
Toolbar	*(Drafting only)* **Annotation**® **Annotation Preferences**
Menu	**Preferences**® **Annotation**
Graphics window	Right-click weld symbol® **Style**
Location in dialog box	**Symbols** tab® **Weld** group® **Weld Standard** **Symbols** tab® **Weld** group® **Weld Symbol Size Factor**

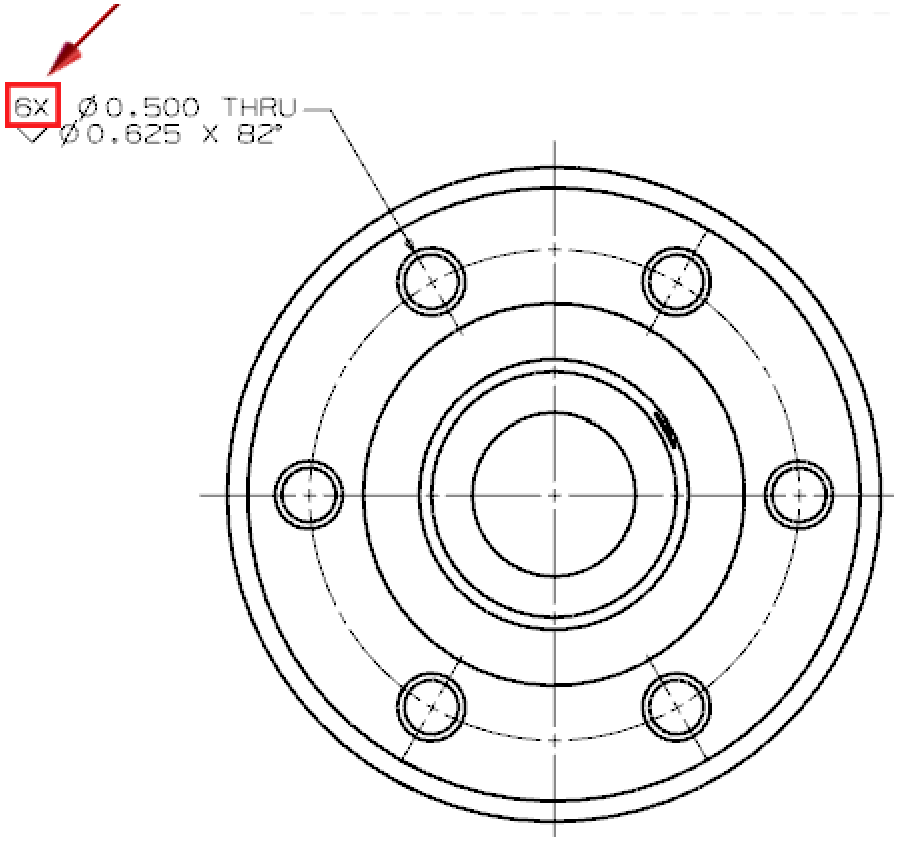
Feature Parameters support for pattern features

When you use the **Feature Parameters** command, you can now automatically create callouts that display the number of instances in a pattern feature. To do this, you can select either:

- The parent feature
- An instance of the pattern feature

You can make your selection from the **Feature Parameters** dialog box or in the graphics window.

Note This enhancement supports all hole types, with the exception of tapered holes.



Where do I find it?

Application	Drafting
Toolbar	**Dimension→Feature Parameters**
Menu	**Insert→Dimension→Feature Parameters**

PMI

PMI user interface enhancements

What is it?

The user interface for the PMI task environment is simplified to improve usability and make the system more intuitive. Enhancements include:

- The **Show** and **Hide** command now supports all PMI annotations.
- The **Center Mark** command is now named **PMI Center Mark**.
- The **3D Centerline** command is now named **PMI Centerline**.
- The **Region** group on the **PMI Region Label** dialog is now named **Settings**.

- The **Inherit** option is added to the **Feature Control Frame**, **Datum Feature Symbol**, **Datum Target**, and **Surface Finish Symbol** dialog boxes

Where do I find it?

PMI Center Mark command

Application	PMI
Toolbar	**PMI**® **Supplemental Geometry Drop-down** list® **PMI Center Mark** 
Menu	**PMI**® **Supplemental Geometry**® **PMI Center Mark**

PMI Centerline command

Application	PMI
Toolbar	**PMI**® **Supplemental Geometry Drop-down** list® **PMI Centerline** 
Menu	**PMI**® **Supplemental Geometry**® **PMI Centerline**

Component PMI in assembly section views enhancement

What is it?

Assembly PMI section views and lightweight section views can now display component level PMI included in PMI assembly filters. The following behaviors support this enhancement:

- At the time of their creation, PMI section views or lightweight section views inherit any PMI assembly filters currently applied to the source assembly model view.

- The list of views available for selection in the PMI assembly filter **Apply to Views** dialog box now includes PMI section views and lightweight section views.

- The list of views available for selection in the **Model View Filter** dialog box, which is used to create PMI assembly filters, now includes PMI section views and lightweight section views in component parts.

Where do I find it?

Apply a PMI assembly filter to a PMI section view

Application	PMI
Prerequisite	You must be in an assembly part
Part Navigator	Right-click **PMI Assembly Filters** node® **Apply to Views**® select PMI section view

Create a PMI assembly filter from a PMI section view

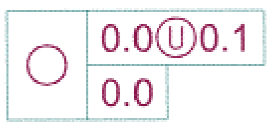
Prerequisite	You must be in an assembly part
Part Navigator	Right-click **PMI Assembly Filters** node® **Add Filter**® **By Model View Name**® select PMI section view

Circle U Tolerance Modifier for feature control frames

What is it?

The Circle U Tolerance Modifier (CUTM) symbol is now available when either the ASME or the ISO GD&T standard is in use.

The CUTM and its corresponding numeric value for Drafting and PMI feature control frames is part of the ASME standard, but is not part of the ISO standard.



Why should I use it?

Your local practice can include the CUTM symbol and value to provide its tolerancing information even if your adopted standard is the ISO standard.

Where do I find it?

Application	Drafting and PMI
Prerequisite	Tolerancing standard is set to ASME or ISO
Toolbar	Drafting toolbar **Feature Control Frame** PMI toolbar **PMI Annotation Drop-down**® **Feature Control Frame**
Menu	**PMI**® **Feature Control Frame** **Insert**® **Annotation**® **Feature Control Frame**

Graphics window	Right-click a feature control frame created in the current application® **Edit**
Location in dialog box	**Frame** group® **Tolerance**® **Tolerance Modifiers**

Find PMI Associated to Geometry

What is it?

- The **Find PMI Associated to Geometry** command can now display results in the Work View by creating a temporary display of the PMI that are associated to the selected geometry. You can zoom and rotate to examine the query results. When you refresh the display or select another view or orientation, the temporary display of the PMI is erased. The temporary display is the default display mode for the command.

  The previous command behavior where a query view is created is still available by means of a new dialog box option, **Create Query Results View**. When that option is selected, NX saves the query results in a model view. The name of the view is changed from **QUERY RESULTS VIEW** to **Query Results**.

- A new preference, **Display Dialog for Find PMI Associated to Geometry from MB3 Menu**, allows you to suppress the display of the dialog box when you start the command from a shortcut menu.

Why should I use it?

- You can display PMI associated to geometry without creating a model view.

- If you want to retain the search results in a model view, you can select the **Create Query Results View** option.

Where do I find it?

Find PMI Associated to Geometry command

Application	PMI
Menu	**Information**® **PMI**® **Find PMI Associated to Geometry**
Graphics window	Right-click geometry® **Find PMI Associated to Geometry**
Location in dialog box	**Search Options** group® **Create Query Results View**

Display Dialog for Find PMI Associated to Geometry from MB3 Menu preference

Menu	**Preferences® PMI® PMI**
Location in dialog box	**Display** tab® **Query Options** group® **Display Dialog for Find PMI Associated to Geometry from MB3 Menu**

PMI leader arrowhead display enhancement

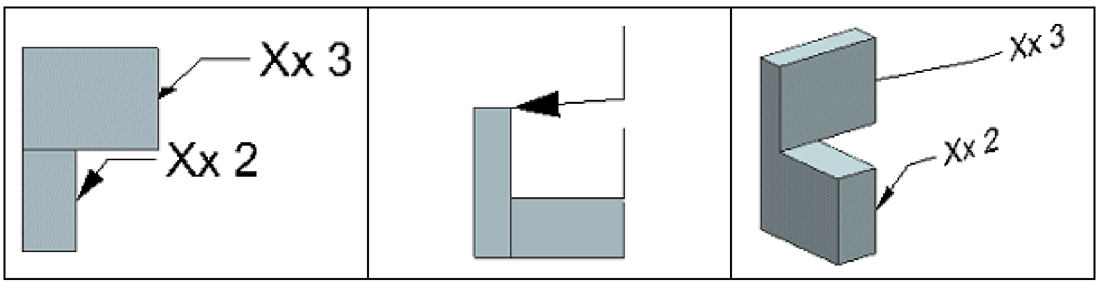
What is it?

PMI arrowheads have been enhanced to provide a more pleasing appearance as the model is rotated.

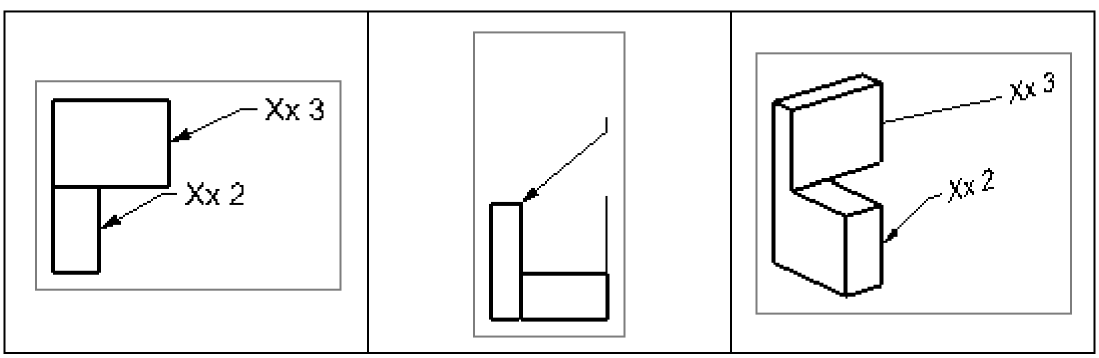
PMI arrowhead behavior now includes the following characteristics and is the same for all arrowhead shapes.

- In the 3D model, leader arrowheads are always drawn in the same plane as the terminating leg of the leader line.

  If the termination point for the leader does not lie in the annotation plane, arrowheads may appear foreshortened in the annotation plane view.



- When PMI is inherited into a drafting view:
  - o If the annotation plane is the same plane as the drafting view, NX rotates leader arrowheads so that they do not appear foreshortened.
  - o If the drafting view is not in the same plane as the annotation plane, the arrowheads remain as they are in the 3D model.



Where do I find it?

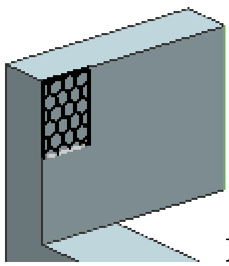
You see the new behavior when creating or editing PMI annotations and symbols.

PMI region crosshatch pattern

What is it?

PMI region functionality is enhanced to make the list of crosshatch patterns that are available in Drafting also available for PMI regions. You can still choose to not apply a crosshatch pattern to the PMI region.

Example



Model including a PMI region with honeycomb crosshatch pattern

Why should I use it?

Use the new crosshatch pattern capability to apply a pattern other than the **General Use** pattern.

Where do I find it?

Display a crosshatch pattern

Application	PMI
Toolbar	**PMI Supplemental Geometry Drop-down**® **PMI Region** 
Menu	**PMI**® **Supplemental Geometry**® **Region**
Location in dialog box	**Settings** group® **Crosshatch** sub-group® **Display Crosshatch**

Select a crosshatch pattern

Application	PMI
Toolbar	**PMI Supplemental Geometry Drop-down**® **PMI Region** 
Menu	**PMI**® **Supplemental Geometry**® **Region**
Location in dialog box	**Settings** group® **Crosshatch** sub-group® **Crosshatch File** **Settings** group® **Crosshatch** sub-group® **Pattern**

PMI support for Geometry Sharing

What is it?

When NX Modeling considers the main part geometry to be shared by geometry in an extracted body, new PMI support for geometry sharing provides the following:

- When you select a PMI node that is associated to main body geometry, NX highlights corresponding geometry on the extracted body.
- Running the **Find PMI Associated to Geometry** command on extracted body geometry finds PMI associated to itself and to its corresponding main body geometry.
- Objects that are highlighted as being associated with PMI based on geometry sharing are reported as *Shared Associated Objects* in the **Dependencies** pane of the Part Navigator, and for commands such as:
  - o **Information**® **Object**
  - o **Information**® **PMI**® **Report**
- When parts are written to JT files, the same PMI-related highlighting behavior for shared geometry occurs in Teamcenter Visualization as it did in NX.

This new PMI capability is enabled or disabled by means of a new preference, **Enable PMI Support for Geometry Sharing**.

Why should I use it?

Use this capability if you want PMI associated with the main geometry for a part to treat shared geometry as if it were the same as the main geometry.

Where do I find it?

Application	PMI
Prerequisites	• Geometry sharing for Modeling (customer default **Share Geometry on Save**) Tip To find a customer default, choose **File→Utilities→Customer Defaults**, and click **Find Default** . • The preference **Enable PMI Support for Geometry Sharing** is selected • Shared geometry has been created by means of the **Extract Body** command • PMI is associated to elements of the main body geometry
Menu	**Preferences® PMI**
Location in dialog box	**General** tab® **Enable PMI Support for Geometry Sharing**

Replace Custom Symbol

Use the **Replace** command to:

- Change the associative link to a master custom symbol for one or more unlocked custom symbol instances. The custom symbol instances can include nested custom symbol instances.

- Repair a link to a master custom symbol.

- Create an associative link to a different master custom symbol.

- Create an associative link from a legacy custom symbol to an existing master custom symbol.

You can update the associativity of PMI custom symbol instances in any application other than Drafting.

Note You cannot replace the associativity of a custom symbol embedded in a text string. However, you can replace all unlocked instances of selected symbols on the current drawing sheet by selecting the **Replace all Instances** option in the **Replace** dialog box. This includes instances embedded in a text string.

If the **Replace all Instances** option is not selected, only the display instances of the selected PMI custom symbol objects will change.

Where do I find it?

Application	PMI
Graphics window	Right-click custom symbol instance® **Replace**

Width options for lines and symbols

What is it?

Additional line width options are available to control the appearance of lines and symbols in NX. All commands and preferences that contain the **Width** or **Line Width** option to define the line or symbol now provide these options.

List of available width options for lines and symbols

The original options of **Thin**, **Normal**, and **Thick** are represented by the **0.13 mm**, **0.18 mm**, and **0.25 mm** options, respectively.

Why should I use it?

Per industry or company standards, more than three line widths may be required when annotating a model or drawing. Varying line widths let you emphasize or de-emphasize geometry and symbols as needed.

Where do I find it?

The width options are available in every dialog box in Drafting and PMI that contain a **Width** or **Line Width** option to define the size of the line or symbol.

Standard font support for symbols

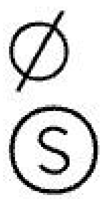
What is it?

When the Drafting or PMI text type is set to a standard font type, symbols embedded in a note or other text are also presented in a font style similar to the text font. If a standard font representation does not exist for a symbol, the symbol is displayed as single-stroked lines. NX font types, such as **blockfont**, still display the symbols as single-stroked lines.

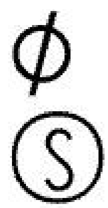
NX 8.0 and earlier

NX 8.5

Symbol font files, which are provided with NX, map the display of a symbol to standard font types. Symbol font files are provided for all of the supported drafting standards, and are set by a **Symbol Font File** annotation preference and customer default.



Example of ANSI symbols



Example of ISO symbols

Where do I find it?

Symbol Font File preference

Application	PMI
Menu	**Preferences® Annotation**
Location in dialog box	**Symbols** tab® **Symbol Font File**

Symbol Font File customer default

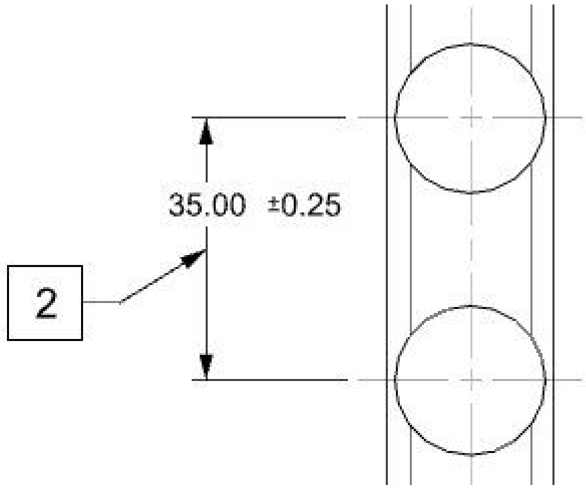
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Drafting® General® Standard** tab® click **Customize Standard® Annotation® Symbols** tab® **Symbol Font File** Note that you must start a new NX session before the change is available.

Leader line enhancements

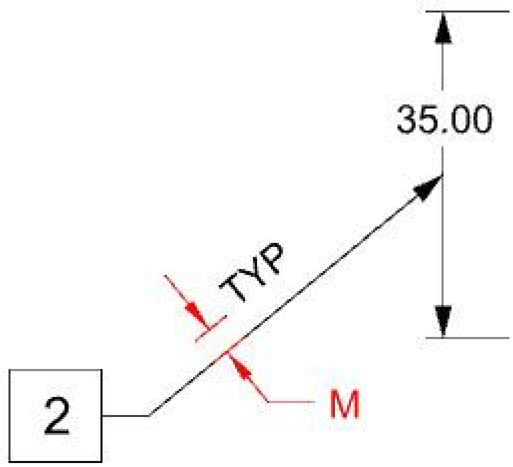
What is it?

Leader lines for Drafting and PMI annotations are improved with the following enhancements:

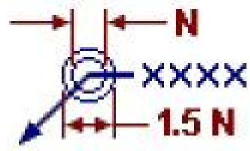
- You can associatively attach an annotation with **Plain** or **All Around** leaders to the midpoint or to a selected point of a dimension line.



- A new spacing option, **M**, lets you control the distance between a leader line and an annotation placed on the leader line using the **Flag** leader type. The spacing is given as a factor of the font size of the annotation you are placing.



- You can control the size of the **All Around** (single circle) and **All Over** (double circle) symbols on a leader line using the new **N** size option. The size is given in inches or millimeters, depending on the unit type of your part. The initial size is equal to the **Character Size** on the **Line/Arrow** tab of the **Annotation Preference** dialog box.



Where do I find it?

Set the **M** and **N** preference or edit the style of an existing leader

Application	Drafting and PMI
Toolbar	*(Drafting only)* **Annotation® Annotation Preferences**
Menu	**Preferences® Annotation**
Graphics window	Right-click leader line® Style
Location in dialog box	**Line/Arrow** tab® **M** **Line/Arrow** tab® **N**

M and **N** customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Drafting® General® Standard** tab® click **Customize Standard**. On the **Annotation® Line/Arrow** tab® **M** and **N**

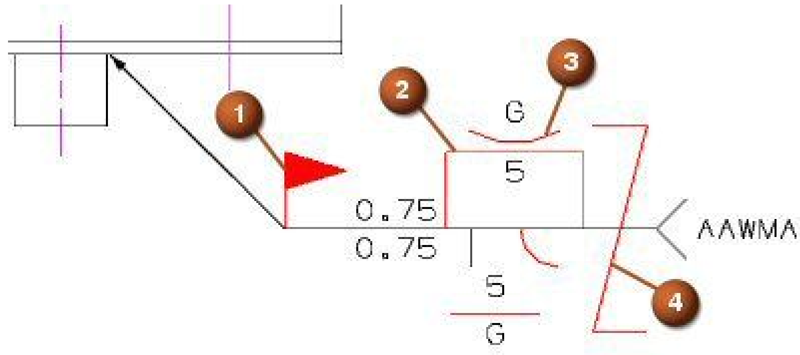
Weld symbol enhancements

What is it?

- A new **Weld Standard** option, available in the **Annotation Style** and **Annotation Preferences** dialog boxes, lets you interactively set the weld standard. This lets you access different standards-driven weld symbols without having to reset the entire drafting standard in your NX session.

- A new **Weld Symbol Size Factor** option lets you control the size of the symbols in a weld symbol. The size is given as a factor of the character size of the symbol, where 1.0 means the symbol is equal to the current character size.

  The size factor is applied to the field symbol (1), the weld symbol (2), the contour symbol (3), and the staggered annotation symbol (4) elements above and below the reference line of the weld symbol.



Where do I find it?

Application	Drafting and PMI
Toolbar	*(Drafting only)* **Annotation**® **Annotation Preferences**
Menu	**Preferences**® **Annotation**
Graphics window	Right-click weld symbol® **Style**
Location in dialog box	**Symbols** tab® **Weld** group® **Weld Standard** **Symbols** tab® **Weld** group® **Weld Symbol Size Factor**

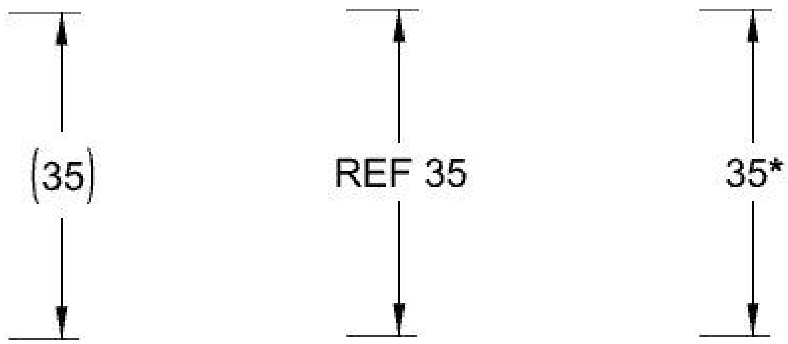
General annotation enhancements

What is it?

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

- Improvements to the selection of intersections and tangent points provide more robust and consistent results for dimension intersections and tangent point associativity.

- New preferences and customer defaults let you specify how to denote a Reference Only dimension.
  - o New **Reference Dimension** preferences let you specify a suffix or a prefix character instead of parentheses to denote a reference dimension. The default is to use parentheses to indicate a reference dimension.



Default **Prefix only** **Suffix only**

  - o Two new customer defaults, **Use 1.5 X Height Parentheses** and **Use Suffix/Prefix Characters**, let you specify the default preference for your NX session.

Reference Dimension preferences with parentheses default

Reference Dimension preferences with characters default

Where do I find it?

Reference Dimension preferences

Application	Drafting and PMI
Toolbar	*(Drafting only)* **Annotation**® **Annotation Preferences**
Menu	**Preferences**® **Annotation**
Location in dialog box	**Dimension** tab® **Precision and Tolerance** group® **Reference Dimension**

Reference dimension customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Drafting® General® Standard** tab® click **Customize Standard**. On the **Annotation® Dimensions** tab, select **Use 1.5 X Height Parentheses** or **Use Suffix/Prefix Characters**. Note that you must start a new NX session before the change is available.

NX Sheet Metal

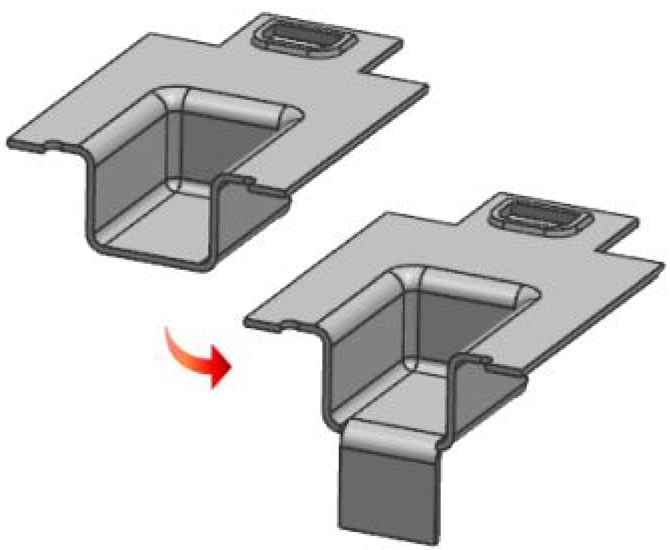
Flange enhancement

What is it?

You can now create a flange on the edge of a deformed face formed by a Dimple feature. The edge of the deformed face must be linear, and the deformed face must be parallel to the original planar face on which the Dimple feature was created.

You can unbend or rebend the flange, or create a Flat Solid feature, or a Flat Pattern feature.

In the example a flange is created on the edge of a deformed face. The deformed face was created by an open section type of Dimple feature.



Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Bend Drop-down** list® **Flange** 
Menu	**Insert**® **Bend**® **Flange**

Pattern Feature in NX Sheet Metal

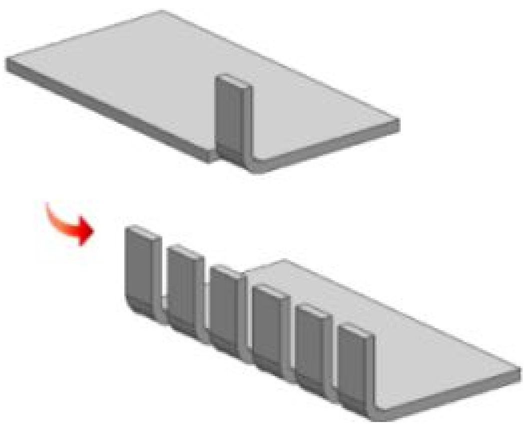
What is it?

The **Pattern Feature** command is now available in the NX Sheet Metal application.

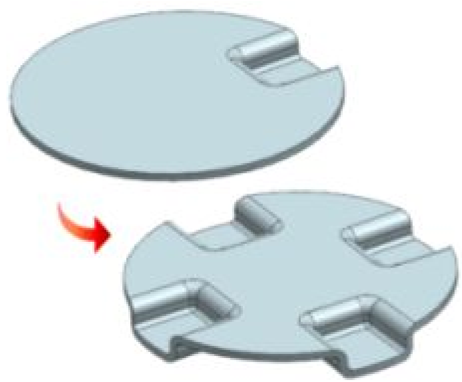
You can create a pattern of the following:

- Tab (primary and secondary).
- Flange, Contour Flange (primary and secondary), Lofted Flange (primary and secondary), Hem Flange, Bend, Jog, Advanced Flange.
- Dimple, Louver, Drawn Cutout, Bead, Normal Cutout, Solid Punch, Gusset.
- Closed Corner, Three Bend Corner, Break Corner, Chamfer, Bend Taper.
- Unbend, Rebend, Unform, Reform.

Note NX Sheet Metal supports only the **Variational** type of pattern method.



Linear pattern of the Flange feature



Circular pattern of the Dimple feature

Why should I use it?

Use this command to fill a face or boundary with a pattern of Sheet Metal features.

Where do I find it?

Application	NX Sheet Metal
Menu	**Insert® Associative Copy® Pattern Feature**

Dimple enhancements

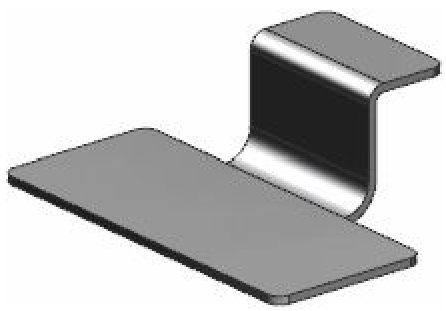
What is it?

You can now create a Dimple feature across cylindrical bends in the unbent state.

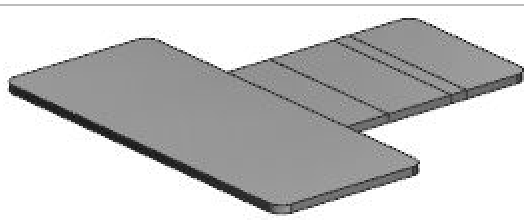
An open or closed section type of the Dimple feature can be created:

- Across a bend and a planar face.
- Completely inside the bend face.

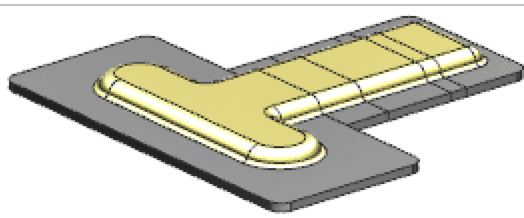
In the following example, an open section type of Dimple feature is created across a bend and a planar face. The Dimple feature is retained after the rebend operation.



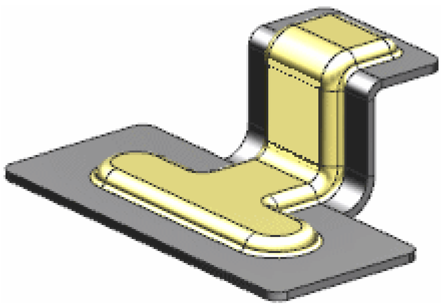
Original part.



Part in unbent state.



Dimple feature created across bend region.



Dimple feature retained after Rebend operation.

Note When the depth direction of the Dimple feature, is the same as the bend direction, the depth of the Dimple feature must be smaller than the bend radius.

Where do I find it?

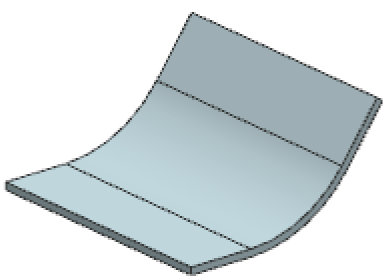
Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Punch Drop-down** list® **Dimple** 
Menu	**Insert**® **Punch**® **Dimple**

Drawn Cutout enhancement

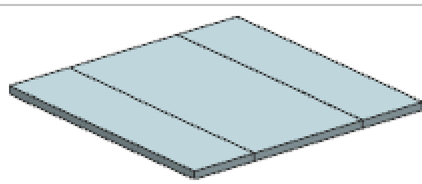
What is it?

You can now create a Drawn Cutout feature across cylindrical bends in the unbent state.

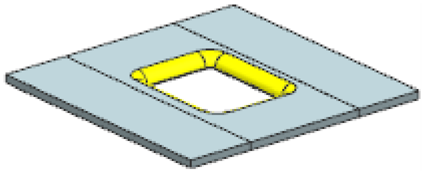
This feature can have an open or a closed section.



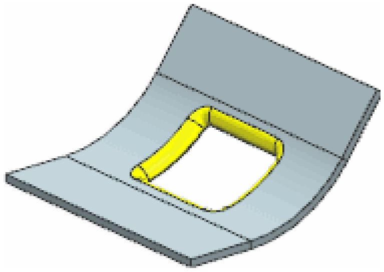
Original part.



Part in unbent state.



Drawn Cutout feature created completely inside the bend region.



Drawn Cutout feature retained after Rebend operation.

Note When the depth direction of the Drawn Cutout feature, is the same as the bend direction, the depth of the Drawn Cutout feature must be smaller than the bend radius.

Where do I find it?

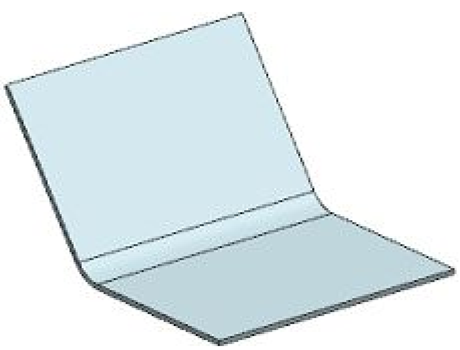
Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Punch Drop–down** list® **Drawn Cutout** 
Menu	**Insert**® **Punch**® **Drawn Cutout**

Bead enhancements

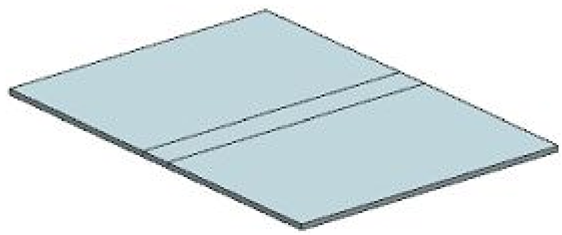
What is it?

Bead feature across bend regions

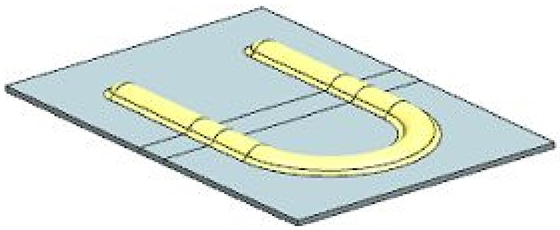
You can now create a Bead feature across bends in the flattened state.



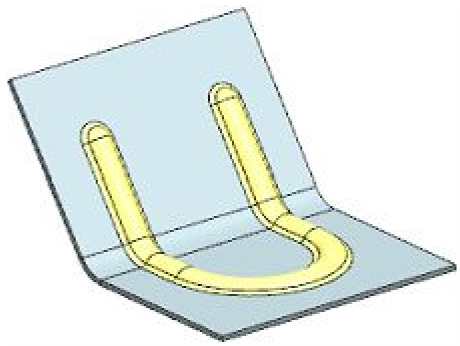
Original part.



Part in unbent state.



Open section type of Bead feature created across bend regions.



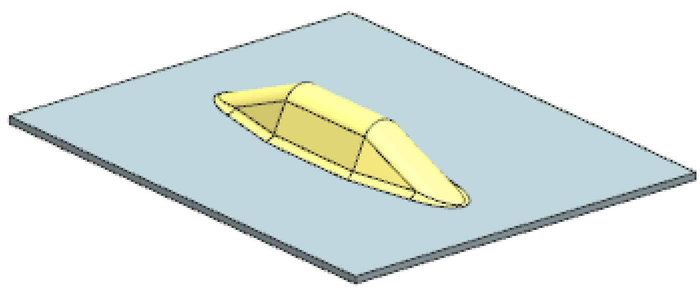
Part rebent along with the Bead feature.

Note When the depth direction of the Bead feature, is same as the bend direction, the depth of Bead feature must be smaller than the bend radius.

Bead feature with a V-shaped cross section

You can now create a Bead feature with a V-shaped cross section and tapered end condition.

The new customer default option, **Taper Distance**, lets you specify the value for the default taper distance for the Bead feature.



Bead feature with a V-shaped cross section and tapered end condition

Where do I find it?

Bead dialog box

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Punch Drop–down** list® **Bead** 
Menu	**Insert**® **Punch**® **Bead**
Location in dialog box	**Bead Properties** group® **Cross Section** list® **V-Shaped** **Bead Properties** group® **End Condition** list® **Tapered**

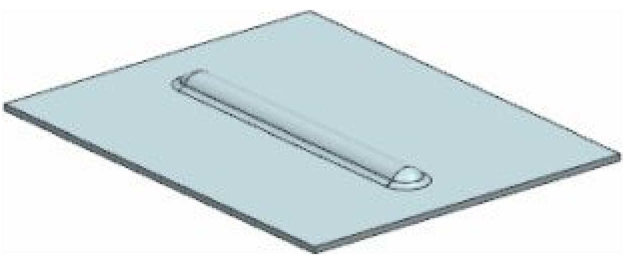
Customer Defaults dialog box

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Sheet Metal**® **Bead**® **General** tab® **Taper Distance**

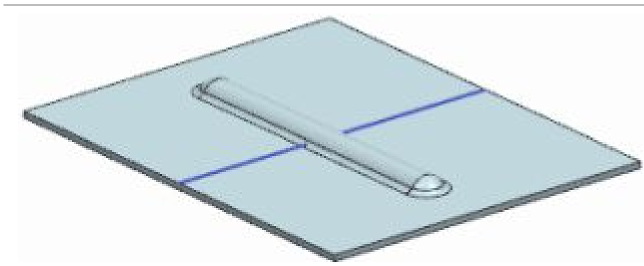
Bend enhancement

What is it?

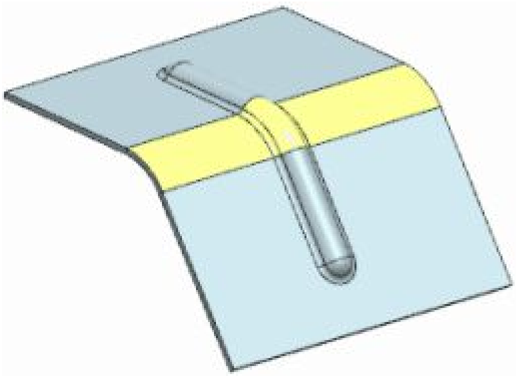
You can now create a Bend feature across existing Bead, Dimple, or Drawn Cutout features.



Tab feature with an existing Bead feature.



Bend line traversing across the Bead feature.



Bend feature created across the Bead feature.

Note When you create such bends, the bend line must completely pass through the deformed features.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Bend Drop–down** list® **Bend** 
Menu	**Insert**® **Bend**® **Bend**

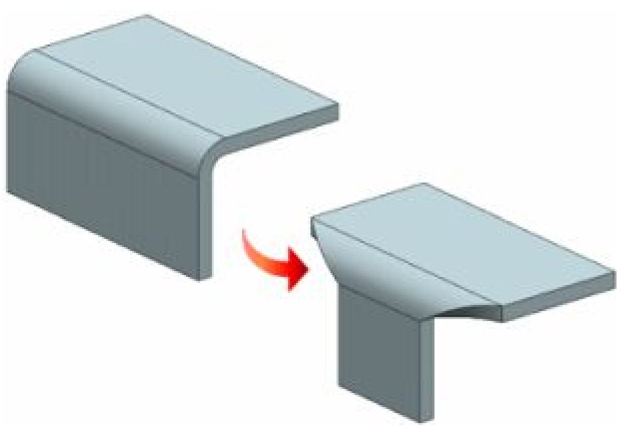
Bend Taper enhancement

What is it?

- When you use the **Bend Taper** command to create tapered bend regions, the following taper options are now available:

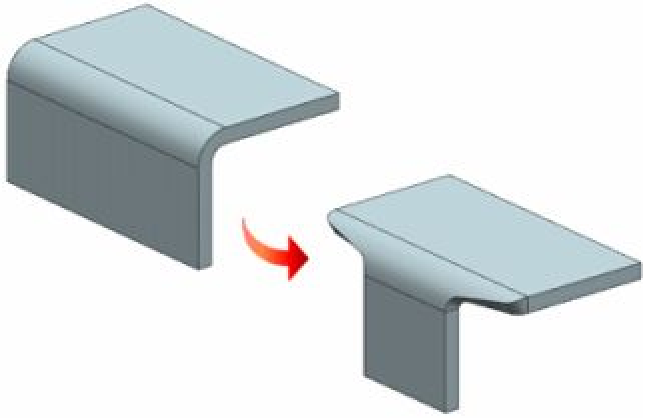
Linear Creates a tapered bend region using the specified taper angle or taper distance value.

This is same as the existing Bend Taper feature. All legacy Bend Taper features will be converted to the **Linear** type with the specified taper angle.

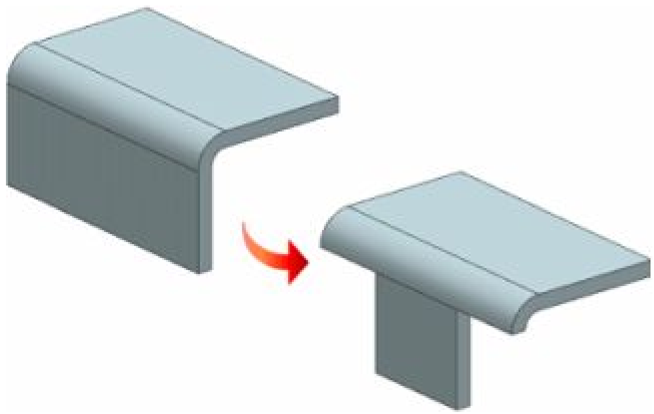


Tangent Creates a tapered bend region that is tangent to the web region, at the specified taper distance.

In the flattened view, the edge of the taper is tangent to the bend region and the web region.



Square Removes material from the web region at the specified distance. No taper is applied to the bend region.



- The **Chaining** options available in the previous **Bend Taper** dialog box, are now replaced with the **Web Taper** options, and renamed as follows:

Chaining options in previous dialog box	Corresponding Web Taper options
Bend Only	**None**
Bend Face	**Face**
Bend Face Chain	**Face Chain**

- You can now specify the taper parameters for the bend region and web region of each taper side using the options available in the **Taper Definition Side 1** and **Taper Definition Side 2** groups.

Why should I use it?

Use these options to remove material from the bend and web regions.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Cut Drop–down** list® **Bend Taper**
Menu	**Insert**® **Cut**® **Bend Taper**
Location in dialog box	**Taper Definition Side 1/Taper Definition Side 2** group® **Bend** sub group® **Taper** list

Convert to Sheet Metal Wizard

What is it?

Use this wizard to rip edges, clean up inconsistencies, and convert a model to a Sheet Metal part.

The following pages are available:

Edge Rip page

Use the options available on this page to rip along the corner edges to convert a solid body to a Sheet Metal body. You can also rip along a linear sketch. You can select the edge, or set of edges, to rip before cleanup.

All the options available on this page are same as the **Edge Rip** dialog box options.

Cleanup Utility page

Use the options available on this page to update the solid body to meet the requirements of the Convert to Sheet Metal feature. You must select a base face to anchor the part during cleanup. You can also define the thickness, sliver tolerance, and specify if you want to show or hide the original body.

All the options available on this page are same as the **Cleanup Utility** dialog box options.

Convert to Sheet Metal page

Use the options available on this page to specify the bend relief parameters and select the base face. If the base face is not selected, the base face selected during the cleanup task is used.

All the options available on this page are same as the **Convert to Sheet Metal** dialog box options.

- The tasks in the **Convert to Sheet Metal Wizard** are optional.
- While the wizard guides you through the workflow, the separate dialog boxes for **Edge Rip**, **Cleanup Utility**, and **Convert to Sheet Metal** commands are also available.

Why should I use it?

The **Convert to Sheet Metal Wizard** helps you perform the cleanup and conversion tasks in the correct sequence.

You can clean up inconsistencies in a model generated by other CAD systems, prior to converting it to a valid NX sheet metal model. The **Convert to Sheet Metal Wizard** streamlines the workflow and ensures a workable Sheet Metal model.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Convert Drop–down** list® **Convert to Sheet Metal Wizard**
Menu	**Insert**® **Convert**® **Convert to Sheet Metal Wizard**

Convert to Sheet Metal enhancement

What is it?

You can now use this command to convert additional geometric conditions like:

- Gussets and ribs.
- Beads, dimples, or drawn cutouts across bend regions.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Convert Drop–down** list® **Convert to Sheet Metal**
Menu	**Insert**® **Convert**® **Convert to Sheet Metal**

Cleanup Utility enhancements

What is it?

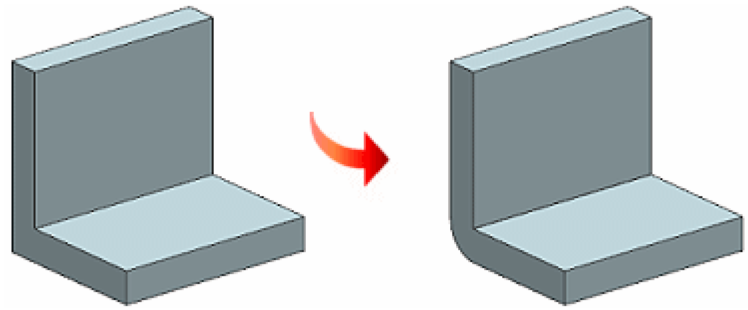
You can now use the **Cleanup Utility** command to:

- Infer the thickness of the resulting Sheet Metal part, after you select the solid body for cleanup.

  If you select the **Infer Thickness** check box, the thickness of the solid body is inferred; else, the user-defined value is used.

- Recognize B-spline faces in a model, and convert them to a simpler form, such as planar or cylindrical.

- Recognize sharp edges in your model, and convert them to bend regions without splitting the thickness face.



Why should I use it?

Use this command to identify and clean up areas of a model that do not meet the requirements of the **Convert to Sheet Metal** command.

You can clean up imported parts and improve successful conversion of non-NX Sheet Metal parts to NX Sheet Metal parts.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Convert Drop–down** list® **Cleanup Utility** 
Menu	**Insert**® **Convert**® **Cleanup Utility**
Location in dialog box	**Thickness** group® **Infer Thickness** check box

Reuse Library

Adding Deformable Components from the Reuse Library

What is it?

When you drag a deformable part from the reuse library to the graphics window, you can now change the value of the deformable parameters as you add it to your model.

Where do I find it?

Resource Bar	Reuse Library
Prerequisite	You must add a deformable component from the Reuse Library to your model.
Dialog Box	The **Deformable Part** dialog box is available after you click **Ok** or **Apply** in the **Add Reusable Component** dialog box.

Add Reusable Component preview window

What is it?

When you add a reusable component from the reuse library to an assembly, you can now view the component in a preview window. You can rotate, pan, or zoom the reusable component in the window before you add it to your model.

Where do I find it?

Resource Bar	Reuse Library
Prerequisite	You must add a reusable component component from the Reuse Library to your model
Location in dialog box	**Display Preview Window** check box.

Adding multiple Reusable Components from the Reuse Library

What is it?

When you add a reusable component from the reuse library to an assembly, you can now add multiple occurrences of the reusable component at specific locations. You can select a reusable component, determine the number of occurrences, and insert each occurrence at a specific location.

Why should I use it?

Using the multiple add capability, you can, for example, select a bolt, washer, and nut from the reuse library and place them multiple times in the assembly at various positions

Where do I find it?

Resource Bar	Reuse Library 
Prerequisite	You must add a reusable component from the Reuse Library to your model.
Location in dialog box	**Quantity in Each Operation** box.

Reusable components in a Teamcenter environment enhancement

What is it?

There is enhanced support for managing reusable components in a Teamcenter environment. You can assign the item ID of an object to a specific set of parameters stored as spreadsheet data. When you set the parameters of a reusable component in the **Primary Parameters** dialog box, the item that is associated with those parameters is added to your model.

In the example below, if a Diameter of 20 is specified as a primary parameter, then item 000345 will be added to the assembly.

PARAMETERS		
Diameter	Length	PART_NUMBER
10	50	000213
20	80	000345
END		

Where do I find it?

Resource Bar	Reuse Library 
Prerequisite	You must add a reusable component from the Reuse Library to your model in a Teamcenter environment.

Part family save directory for reusable components enhancement

What is it?

You can now assign a system folder in which to store individual reusable components as part family members using the **Family Member Save Directory** box. It was previously only possible to specify this folder through the customer defaults.

Where do I find it?

Resource Bar	Reuse Library
Dialog box	**Create/Edit KRX file**

Product Template Studio enhancements

What is it?

As the result of these enhancements, you can now do the following:

- When you create a list of options in a template dialog box for a number, string, and integer field, you can use an NX list expression for the displayed options in the template. The list expression can read values from multiple dynamic resources such as a spreadsheet.

- Have a visual rule that invokes a journal script generated in NX. Compiling the code into a DLL file is no longer required.

  Note This visual rule can only execute .vb or .cs scripts. It cannot execute grip, java or C++.

- Use a formula in the integer, number, and string fields when you create a template.

- Check requirement results are now displayed as alerts instead of inside the information window.

Where do I find it?

Application	Product Template Studio

Routing

Configuring Routing applications

Multiple nodes for part type in PTB files

What is it?

You can now specify multiple nodes for part types in PTB files.

Example In *ugroute_mech_metric.xml* / *ugroute_mech_inch.xml*, you can specify multiple nodes for a part type using the following code:

```
<VALVE_PARTS_NODE>
DIN_PIP_VALVES_1;DIN_PIP_VALVES_2;DIN_PIP_VALVES_3
</VALVE_PARTS_NODE>
<ELBOW_PARTS_NODE>
DIN_PIP_ELBOW_1;DIN_PIP_ELBOW_2
</ELBOW_PARTS_NODE>
<REDUCER_PARTS_NODE>
DIN_PIP_REDUCER
</REDUCER_PARTS_NODE>
```

When you specify multiple nodes for a part type, NX searches the specified nodes for defining a part as a specified type. NX searches the nodes in the following order:

1. COMPENSATOR
2. MEASUREMENT
3. VALVE
4. PUMP
5. ELBOW
6. GASKET
7. SCREW_STRAIGHT
8. SCREW_ANGLE
9. SCREW_T
10. REDUCER
11. STUBEND
12. TEE
13. FLANGE
14. NOZZLE
15. MOUNTING
16. MEASUREMENT_HOLDER

If NX does not find the part type, it is set to MISCELLANEOUS.

If you specify the same node type against different part types, then the first part type it was listed against in the list is used to set the part type.

Example If you set the DIN_PIPE_XYZ node as the node for both VALVE and ELBOW, then the part type is set to VALVE, because the VALVE node is processed first.

Routing Systems

Place Part enhancements

What is it?

- NX displays an alert when a part is placed incorrectly. For example, when the placement of a fitting breaks the continuity of the flow

- The first part placement solution provided by NX is now based on the PREFERRED_PORT or CROSS_SECTION attributes defined on the routing ports of a part, in that order, after the flow direction and part type processing is done.

- The placement of mountings and screw-seats is automatically updated when you change the dimensions of the stock or when you replace a fitting with another of the same part type. NX deletes the placement and displays an alert if the mounting or screw-seat does not lie on the stock or fitting it was placed on after the edit. If you use the **Undo** command to revert the change, the placement of the mounting or screw-seat reverts as well.

- You can specify required and optional attributes through the **Attributes** dialog box when placing parts. Use the **Show Attribute Dialog on Place Part** customer default to make the **Attributes** button available on the **Place Part** dialog box.

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

Where do I find it?

Application	Routing
Toolbar	Routing toolbar® **Part Drop-down** list® **Place Part**
Menu	**Insert® Routing Part® Place Part**
Resource bar	**Reuse Library**® Drag the part you want to place to the graphics screen and drop it at the location you want to place the part at.

Attributes for bend corner

What is it?

You can now predefine a list of attributes for a bend corner and select from this list when you use the **Assign Corner** command to create a bend corner. You can use templates to predefine the attributes or you can import the attributes from an XML file into the templates.

Tip To create templates choose **File**® **Utilities**® **Attribute Templates**.

Where do I find it?

Application	Routing
Toolbar	Routing toolbar® **Path Drop-down** list® **Assign Corner** 
Menu	**Insert**® **Routing Path**® **Assign Corner**
Shortcut menu	Right-click an RCP® **Assign Corner**

Mirror Routing objects

What is it?

Use the **Mirror Assemblies** command to mirror Routing objects. Examples of Routing objects include ports and built in paths, linear and spline segments, legacy stock, and stock as components.

When you mirror a Routing assembly, NX:

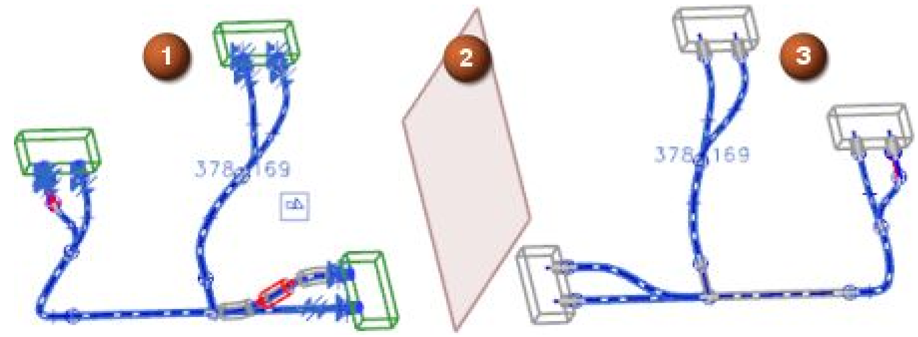
- Creates non-associative mirror objects from the original Routing objects. You can continue to set non-routing mirrored objects to be either associative or non-associative.

  Note Ports and built-in paths of the components are non-associative in the mirrored assembly even when the **Type** is set to **Associative**.

- Mirrors default constraints.

- Mirrors attributes as specified by the APV file configuration.

Note Mirroring segments with ribbon cable stock is not supported.



1	Original Routing assembly
2	Mirroring plane
3	Mirrored assembly with paths, stock, and parts

Why should I use it?

You can use this command to create a symmetric routing network.

Where do I find it?

Application	Routing
Prerequisite	You must select the **Assemblies** check box on the **Start** menu.
Toolbar	**Assemblies**® **Components Drop-down** list® **Mirror Assembly**
Menu	**Assemblies**® **Components**® **Mirror Assembly**

Run Navigator enhancements

What is it?

New commands and options are now available.

Create Run command

Use this command to create a run with specifications. You can create runs with as little information as you want. You can edit runs to add more information.

Example You can create a run in one of these ways:

- Specify only the **Run Identifier** and then edit it to add **From Items**, **To Items**, and **Member Items**.

- Use components with **Reference IDs** or even **Virtual Reference Identifiers** for **From Items** or **To Items** or both.

Find by Attribute command	Use this command to search for runs and items of a run, based on attribute names and attribute values.
Assign Path command	Use this command to assign routing objects, such as paths, RCPs, or stock, to a run. You can also use this command to assign member objects to a run.
Make Active Run, **Make Inactive Run** options	Use these options to make an inactive run active and an active run inactive. Right-click an inactive run in the **Run Navigator** and choose **Make Active Run** to make it active and set its specification as the active one in the **Reuse Library**. Any routing objects placed after that are included in the active run. Right-click an active run in the **Run Navigator** and choose **Make Inactive Run** to make it inactive and set the active specification to **None** in the **Reuse Library**.
Flip Component command	Use this command to flip a component in order to match the flow direction of the component to the flow direction of the run.
Flow direction	When you create a run without specifying the **From Items** and **To Items**, NX fixes the flow direction based on the timestamp order of the start and end segments of the run. The run flows from the older segment to the newer segment. **Tip** To reverse the flow direction, right click the run and choose **Reverse Flow**.

Where do I find it?

Run Navigator commands

Application	Routing Mechanical, Routing Logical
Run Navigator	Right-click and choose **Create Run/Find by Attributes**
Menu	**Tools® Run Navigator® Create Run/Find by Attributes**

Run shortcut menu commands

Application	Routing Mechanical, Routing Logical
Run Navigator	Right-click a run and choose **Assign Path/Make Active Run/Make Inactive Run** Right-click a component in a run and choose **Flip Component**

Unify Path enhancements

What is it?

The **Unify Path** command is enhanced to identify parts according to their specifications. If you select multiple objects that belong to the same run, NX uses the run specifications to find replacement parts.

Note NX does not replace a part that has the specification attribute **Overriden**. To replace such a part, use the **Edit**® **Routing Part**® **Replace Part** command.

Where do I find it?

Application	Routing
Toolbar	Routing toolbar® **Edit Path Drop-down**® **Unify Path** 
Menu	**Edit**® **Routing Path**® **Unify Path**

Routing Mechanical

Handrail Creator

What is it?

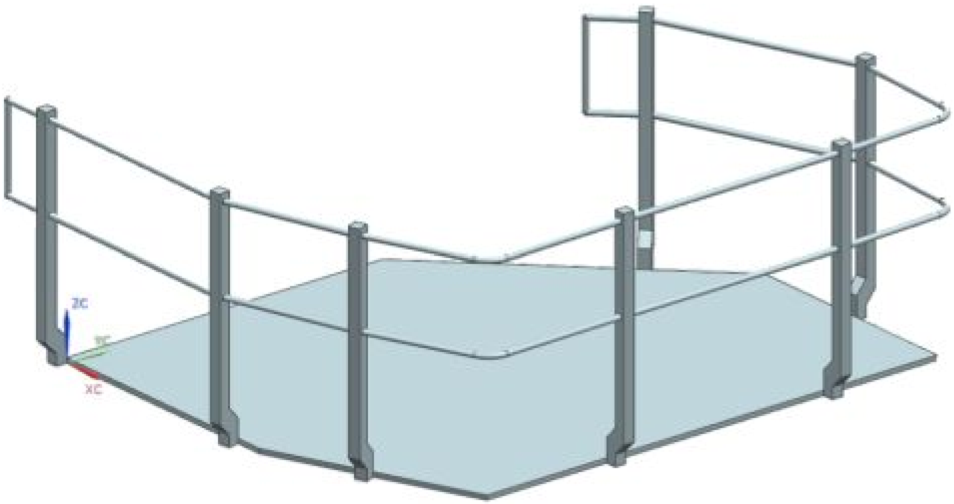
Use this command to build handrails. Each handrail is a subassembly of linear and spline paths, standard parts, and stock.

To use this command, you must create:

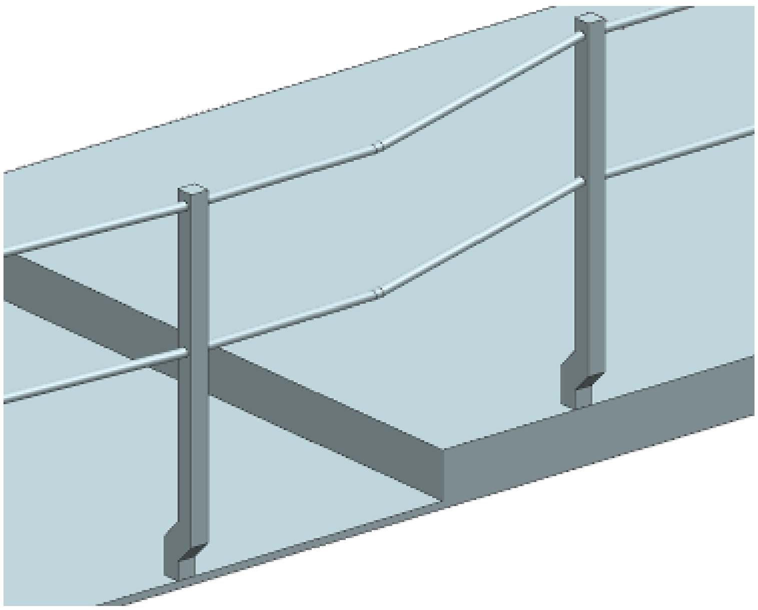
- A Reuse Library of railing stock and handrail posts.
- Part Table files (PTB files) that define the following:
  - o A Start post, which is the post that is placed at the start of the handrail.
  - o An End post, which is the post that is placed at the end of the handrail.
  - o An Intermediate post, which is the post that is placed between the Start and End posts.

Note The PTB files are referenced from the Routing Mechanical Part Library View (PLV) file.

The following example shows handrails on a platform.



The following example shows handrails with bend corners that accommodate the change in elevation.



Why should I use it?

Use this command to easily build handrails on platforms.

Where do I find it?

Application	Routing Mechanical
Toolbar	**Routing Mechanical**® **Tools Drop-down** list® **Handrail Creator**
Menu	**Tools**® **Handrail Creator**

Transform Path enhancement

What is it?

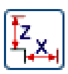
You can now use the **Along Curve-Angle** option in the **Transform Path** command to move or copy a part along a specified curve using linear and rotation handles at the same time.

Where do I find it?

Application	Routing Mechanical
Toolbar	**Routing Mechanical**® **Edit Path Drop-down** list® **Transform Path** 
Menu	**Edit**® **Routing Path**® **Transform Path**
Shortcut menu	Right-click the object to copy or move® **Transform Path**
Location in dialog box	**Transform** group® **Motion** list® **Along Curve-Angle**

Shipbuilding

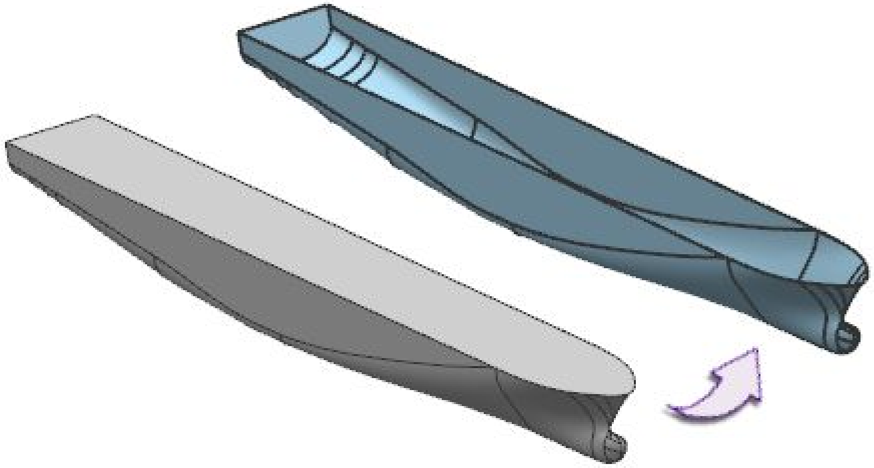
Basic Design



Hull

What is it?

Use the **Hull** command to create the hull of the ship by specifying existing sheet bodies or faces, the exterior boundary, and stock information. The hull is a feature represented by a single sheet body in a separate design element or component.



Hull created using the faces of a solid body

You can:

- Select planes, faces, sheet bodies, or curves to specify the exterior boundary. You can define multiple closed boundaries.
- Apply the thickness inboard, outboard, or centered from the selected sheet bodies and faces.
- Specify material, stock information and other custom attributes attached to the faces of the resulting sheet body.
- Apply welding characteristics to the outer edges of the sheet body.
- Modify the thickness, material, and tightness later when you split the hull into subsystems.

Why should I use it?

Use this command to define the model and attribute information for the hull. This information is recognized by other commands and can be used later when you split the hull into subsystems and transition to the detail design.

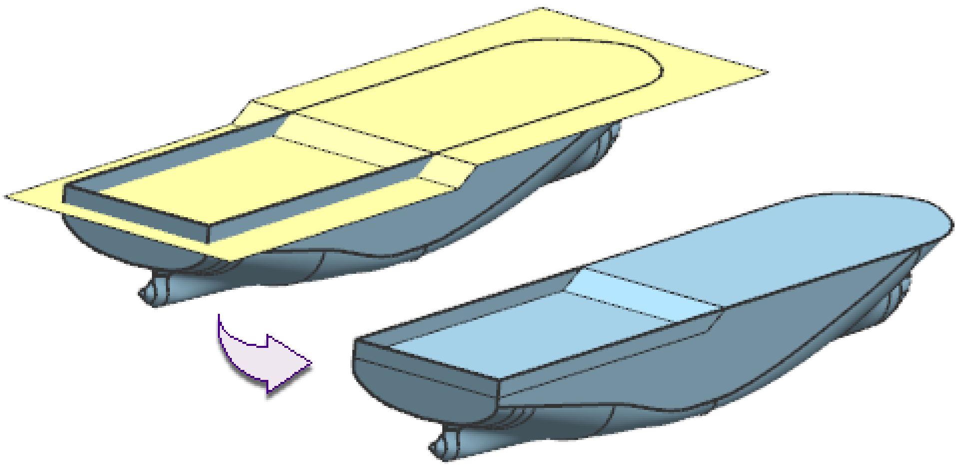
Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Hull**
Menu	**Insert® Steel Features® Hull**

Deck

What is it?

Use the **Deck** command to define a deck in the ship by specifying a mold face, the exterior boundary, and stock information. Each deck is a feature represented by a single sheet body in a separate design element or component.



Deck created using a sheet body and bounded by the hull

You can:

- Specify a round camber, a straight camber, or select an existing sheet body or planes to define the mold face.
- Select planes, faces, sheet bodies, or project curves to specify the exterior boundary.
- Apply the thickness direction up, down, or centered from the specified mold face.
- Specify material, stock information and other custom attributes attached to the faces of the resulting sheet body.
- Apply welding characteristics to the outer edges of the sheet body.
- Specify the tightness of the deck.
- Modify the thickness, material, and tightness later when you split the deck into subsystems.

Why should I use it?

Use this command to quickly define the model and attribute information for decks. This information is recognized by other commands and can be used later when you split the decks into subsystems and transition to the detail design.

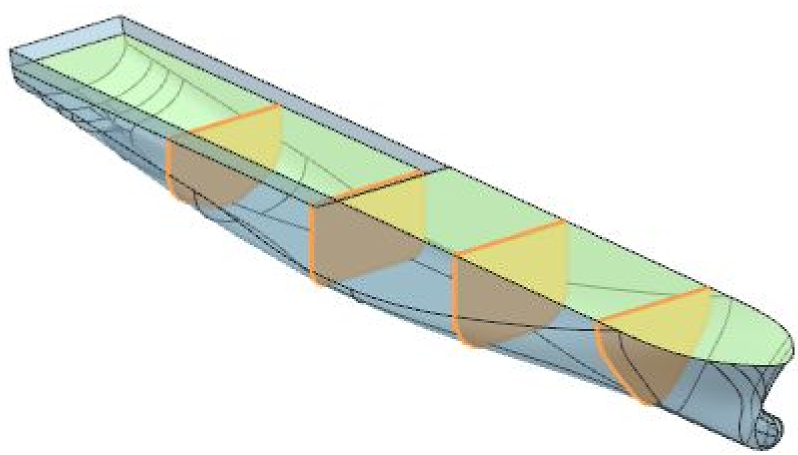
Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Deck**
Menu	**Insert® Steel Features® Deck**

Transverse Bulkhead

What is it?

Use the **Transverse Bulkhead** command to define a bulkhead which is roughly normal to the X-axis of the ship coordinate system. Each bulkhead is a feature represented by a single sheet body in a separate design element or component.



Multiple transverse bulkheads bounded by the hull and a deck

You can:

- Select a plane or sheet body to define the mold face.
- Select planes, faces, sheet bodies, or curves to specify the exterior boundary. You can define multiple closed boundaries.
- Apply the thickness direction aft, forward, or centered from the specified mold face.

- Specify material, stock information and other custom attributes attached to the faces of the resulting sheet body.
- Apply welding characteristics to the outer edges of the sheet body.
- Specify the tightness of the bulkhead.
- Modify the thickness, material, and tightness later when you split the bulkhead into subsystems.

Why should I use it?

Use this command to quickly define the model and attribute information for transverse bulkheads. This information is recognized by other commands and can be used later when you split the bulkheads into subsystems and transition to the detail design.

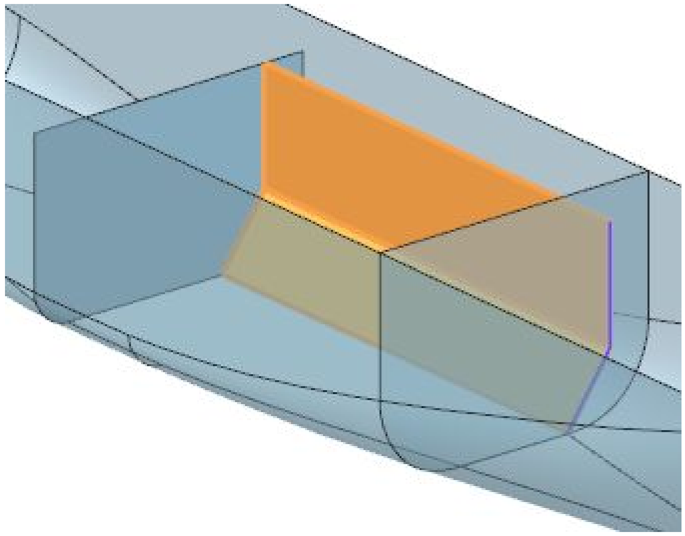
Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Transverse Bulkhead** 
Menu	**Insert® Steel Features® Transverse Bulkhead**

Longitudinal Bulkhead

What is it?

Use the **Longitudinal Bulkhead** command to define a bulkhead which is roughly normal to the Y-axis of the ship coordinate system. Each bulkhead is a feature represented by a single sheet body in a separate design element or component.



Longitudinal bulkhead defined by extruded sketch

You can:

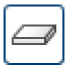
- Select a plane, select a sheet body, or extrude curves to define the mold face.
- Select planes, faces, sheet bodies, or curves to specify the exterior boundary. You can define multiple closed boundaries.
- Apply the thickness direction inboard, outboard, centered, port, or starboard from the specified mold face.
- Specify material, stock information and other custom attributes attached to the faces of the resulting sheet body.
- Apply welding characteristics to the outer edges of the sheet body.
- Specify the tightness of the bulkhead.
- Modify the thickness, material, and tightness later when you split the bulkhead into subsystems.

Why should I use it?

Use this command to quickly define the model and attribute information for longitudinal bulkheads. This information is recognized by other commands and can be used later when you split the bulkheads into subsystems and transition to the detail design.

Where do I find it?

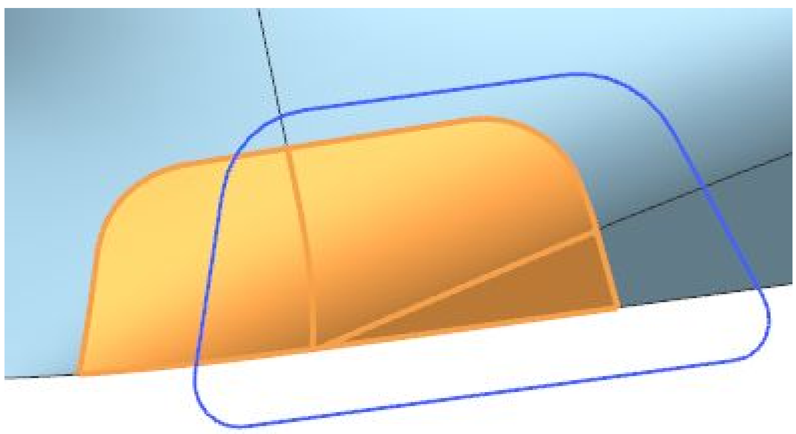
Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Longitudinal Bulkhead** 
Menu	**Insert® Steel Features® Longitudinal Bulkhead**



Generic Plate System

What is it?

Use the **Generic Plate System** command to define a system of plates that are not categorized as a hull, deck, transverse bulkhead, or longitudinal bulkhead system. Each plate system is a feature represented by a single sheet body in a separate design element or component.



Generic plate system defined by projecting a sketch on a sheet body

You can:

- Select a plane or a sheet body to define the mold face.
- Select planes, faces, sheet bodies, or curves to specify the exterior boundary. You can define multiple closed boundaries.
- Apply the thickness direction inboard, outboard, or centered from the specified mold face.
- Specify material, stock information and other custom attributes attached to the faces of the resulting sheet body.
- Apply welding characteristics to the outer edges of the sheet body.
- Specify the tightness of the plate system.

- Modify the thickness, material, and tightness later when you split the plate system into smaller subsystems.

Why should I use it?

Use this command to define the model and attribute information for a plate system that is not categorized as a hull, deck, or bulkhead. The generic plate system can be recognized by other commands that require a ship structure plate system as input.

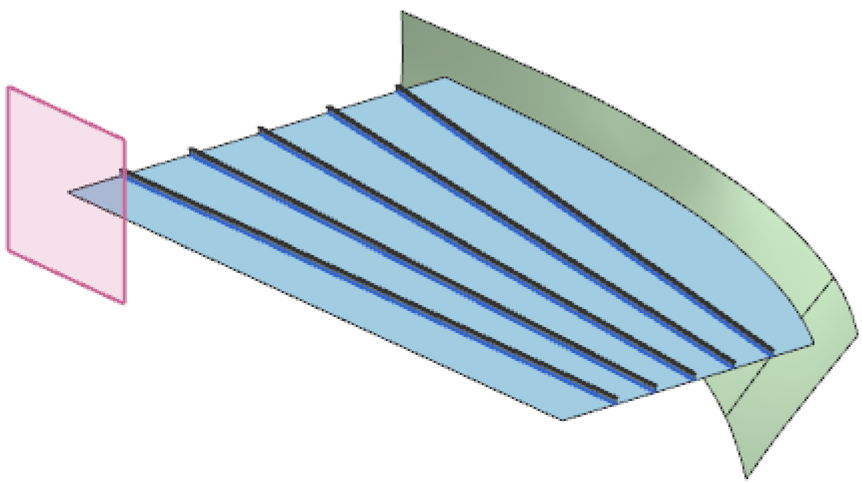
Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Generic Plate System**
Menu	**Insert® Steel Features® Generic Plate System**

Stiffener System

What is it?

Use the **Stiffener System** command to quickly generate stiffener systems on existing ship structures. Each stiffener system is a feature represented by curves in the same design element or component as the selected structure.



Stiffener system created using offset planes

You can:

- You can create multiple stiffener systems on multiple ship structures at one time.
- Specify a section type and stock information from a spreadsheet-driven library. By default, NX provides basic section types.
- Select multiple sets of connected curves, planes, offset planes, or points to define the stiffener paths. You can also specify boundaries for the paths.
- Specify end cuts to the start and end of the stiffener system. **Connected**, **Flange Free**, and **Sniped** end cut types are available.
- Apply optional welding characteristics.

Why should I use it?

Use this command to quickly define the model and attribute information for multiple stiffener systems on structural plate systems. This information is recognized by other commands and can be used later when you transition to the detail design.

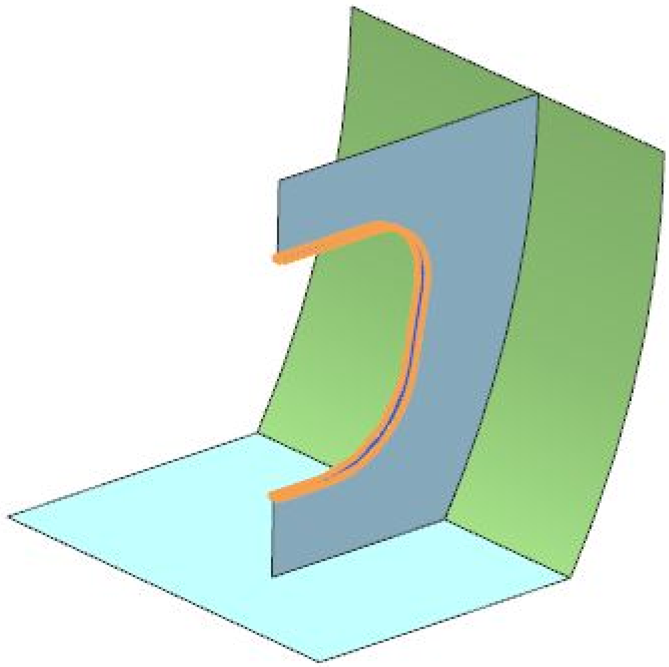
Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Stiffener System**
Menu	**Insert® Steel Features® Stiffener System**

Edge Reinforcement System

What is it?

Use the **Edge Reinforcement System** command to create structural members on the free edges of existing plate systems. Each edge reinforcement system is a feature represented by curves in the same design element or component as the selected plate system.



You can:

- You can create multiple edge reinforcement systems on multiple ship structures at one time.

- Specify a section type and stock information from a spreadsheet-driven library. By default, NX provides basic section types.

- Specify limiting ship structures and define an orientation.

- Specify boundaries for the edge reinforcement system.

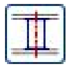
- Apply optional welding characteristics.

Why should I use it?

Use this command to quickly define model and attribute information for edge reinforcements to strengthen the free edges of plate systems. This information is recognized by other commands and can be used later when you transition to the detail design.

Where do I find it?

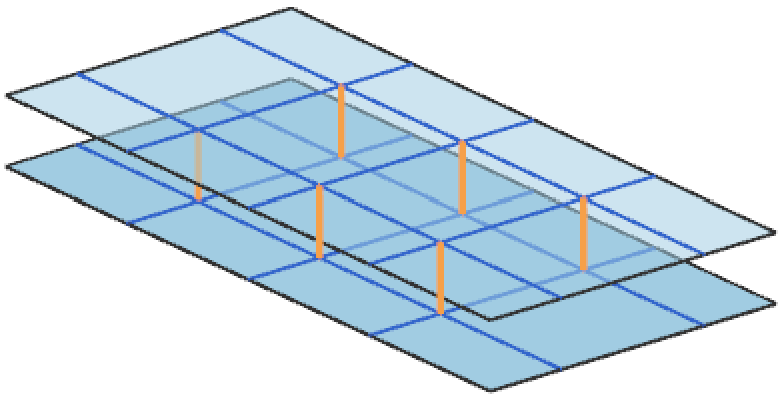
Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design**® **Edge Reinforcement System**
Menu	**Insert**® **Steel Features**® **Edge Reinforcement System**



Pillar System

What is it?

Use the **Pillar System** command to create support structures having a standard sectional shape between two other structures. Each pillar system is a feature represented by curves.



Pillar systems between two decks at stiffener intersections

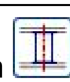
You can:

- You can create multiple pillar systems at one time.
- Specify a section type and stock information from a spreadsheet-driven library. By default, NX provides basic section types.
- Specify the limits of the pillar system by selecting existing ship structures or points.
- Define pillar systems at the intersections of existing ship structures.

Why should I use it?

Use this command to quickly define multiple support structures between two existing structures. Typically, these include vertical structures between decks.

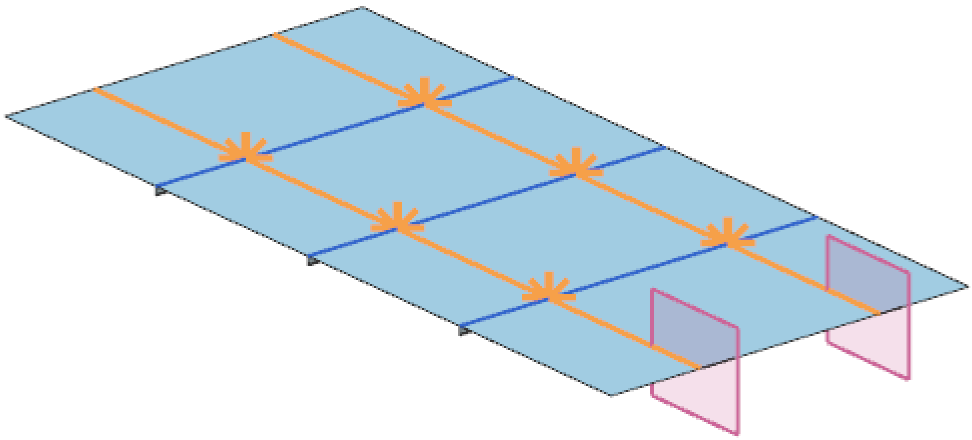
Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Pillar System** 
Menu	**Insert® Steel Features® Pillar System**

Seam

What is it?

Use the **Seam** command to create seams in existing ship structures. Each seam is a feature and is created in the same design element or component as the selected structure. Seam features are represented by curves on decks, bulkheads, and generic plate systems and represented by points on stiffener and edge reinforcement systems.



Seams on a deck and stiffener systems

You can:

- Specify the following types of seams:
  - o **Scantling**
  - o **Erection**
  - o **Straking**
  - o **Intersection**

- Control which types of seams split the selected structures.

- Specify welding characteristics for the seams.

Why should I use it?

Use this command to define seams to split plate and profile systems into smaller subsystems.

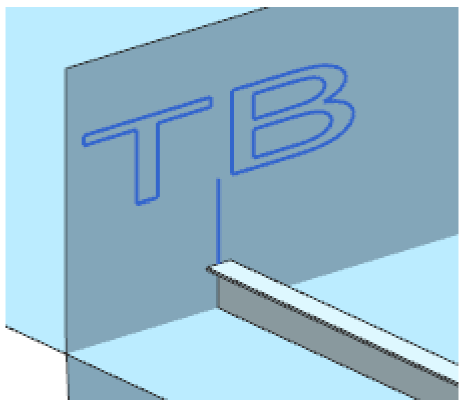
Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Seam**
Menu	**Insert® Steel Features® Seam**

Add Standard Parts enhancements

What is it?

The **Add Standard Parts** command is enhanced to support the creation of standard parts in the Ship Structure Basic Design application.



Tripping bracket added as standard part

- The standard part is represented as a curve. You can use the **Display Solid** command to display a solid body representation.

- A standard part feature is created in the same component or design element as the object containing the selected placement face.

- You can specify the types of standard parts and the parameters of standard parts in a spreadsheet. NX provides templates for tripping brackets, stiffener end brackets, and collar plates.

- You can define the user interaction and behavior for positioning the standard part in basic design in the configuration xml file.

Why should I use it?

You can add standard ship structural parts to a basic design displayed as simplified representations and transition them to the detail design.

Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Add Standard Parts**
Menu	**Insert® Steel Features® Add Standard Parts**

Transition Parts

What is it?

Use the **Transition Parts** command to convert basic design features, including standard parts, into detail design features.

You can:

- Specify the scope of the transition by selecting existing ship sections or plate systems.
- Divide plate and stiffener systems at defined seams and intersections.
- Create new detail structures or update existing structures that have already been converted.

Why should I use it?

This command lets you use the data created for the basic design to define the detail design for the ship.

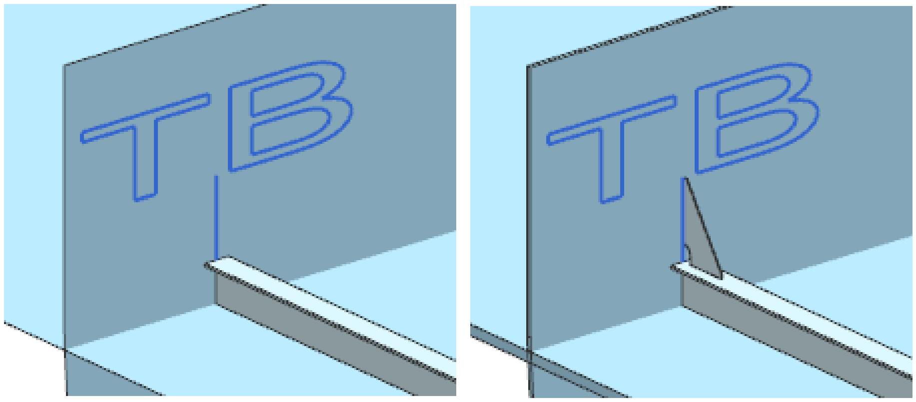
Where do I find it?

Application	Ship Structure Basic Design
Toolbar	**Ship Structure Basic Design® Steel Feature Drop-down** list® **Transition Parts**
Menu	**Insert® Steel Features® Transition Parts**

Display Solid

What is it?

Use the **Display Solid** command to control the display of selected ship structures. You can display structures as curves or as curves with solid bodies.



Tripping bracket displayed as curves and as curves with a solid body

Why should I use it?

This command makes it easier to display basic design ship structures as solid bodies for more realistic representations or as curves to simplify the graphics window.

Where do I find it?

Application	Ship Structure Basic Design
Menu	**View® Display Solid**
Graphics window	Right-click ship structure object® **Display Solid**

Edit Welding Characteristics

What is it?

Use the **Edit Welding Characteristics** command to modify the weld characteristics associated with selected edges or curves of seams, plates, and stiffeners.

You can:

- Select multiple curves and edges from different ship structures.
- Inherit the weld characteristics of an existing curve or edge and apply them to the selected objects.

The weld characteristics are defined as attributes that are attached to the curves and edges. You can customize them for different types of ship structures in the customer defaults.

Why should I use it?

Use this command to edit the weld characteristics of many objects at once instead of editing them individually. This information is attached to the curves and edges and can be used later by other commands.

Where do I find it?

Application	Ship Structure Basic Design
Menu	**Edit**® **Steel Features**® **Edit Welding Characteristics**

Weld Characteristics customer defaults

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Ship Design**® **Basic Design**® Select the tab for the appropriate ship structure type

Edit Stock

What is it?

Use the **Edit Stock** command to modify the stock information for selected areas of plate systems and segments of profile systems that have been created by scantling, intersection or erection seams.

- You can select multiple objects from different ship structures.
- The stock information is defined as attributes that are attached to the selected faces.

Why should I use it?

Use this command to edit the stock of many ship structures at once instead of editing them individually. This information is attached to the selected objects and can be used later by other commands.

Where do I find it?

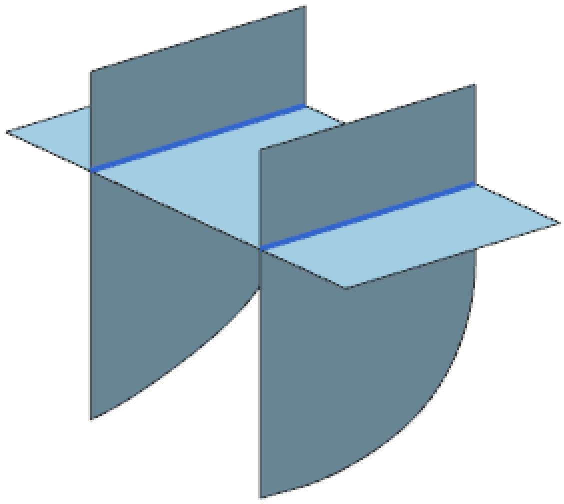
Application	Ship Structure Basic Design
Menu	**Edit**® **Steel Features**® **Edit Stock**

Reverse Split

What is it?

Use the **Reverse Split** command to change the split behavior of intersection seams.

For example, if you initially use the **Seam** command to split a deck with bulkheads, you can use this command to select the seams and reverse the behavior so the bulkheads are split by the deck. The intersection seams are removed from the deck and added to the bulkheads.



Why should I use it?

Use this command to quickly change the behavior of multiple intersection seams without having to manually remove the seams and add them to different ship structures.

Where do I find it?

Application	Ship Structure Basic Design
Menu	**Edit® Steel Features® Reverse Split**

Delete Seam

What is it?

Use the **Delete Seam** command to delete existing seams from ship structures.

You can select multiple seams from different ship structures to delete at one time.

Why should I use it?

This command saves time since you do not have to edit each ship structure to remove each seam individually.

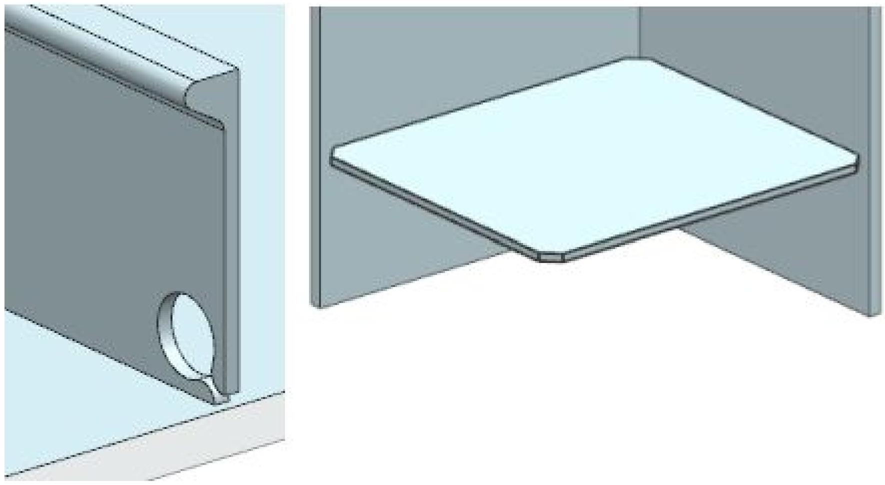
Where do I find it?

Application	Ship Structure Basic Design
Menu	**Edit® Steel Features® Delete Seam**

Corner Cut

What is it?

Use the **Corner Cut** command to create cuts with a predefined shape at the corners of plates or stiffeners. These cuts may be required to provide clearance for weld fillets, access to create a weld, drainage, or stress relief.



You can:

- Specify a section type from a spreadsheet-driven sketch library. By default, NX provides basic section types.
- Select specific corner points or select a plate to infer multiple corner points.
- Select a seam curve to define the size of an elongated corner cut.

Why should I use it?

You can quickly create cuts at the corners of plates and stiffeners for various purposes.

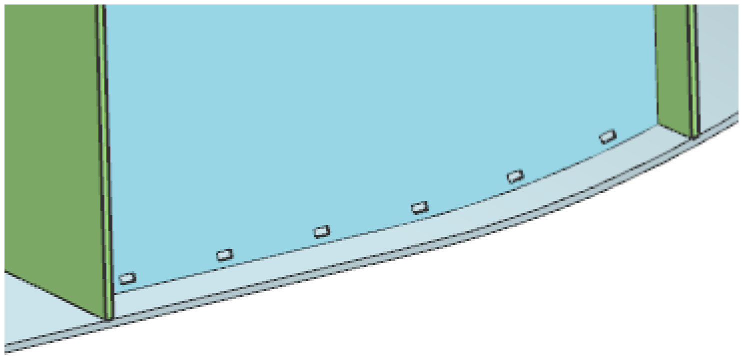
Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Corner Cut**
Menu	**Insert® Steel Features® Corner Cut**

 Edge Cut

What is it?

The **Edge Cut** command replaces the **Ventilation Holes** command. Use this command to create a specified number of cuts along an edge or curve for drainage or ventilation.



You can:

- Specify a section type from a spreadsheet-driven sketch library. By default, NX provides basic section types.
- Define an orientation angle and offset from the selected edge or curve.
- Select start and end limits if you do not want the cuts along the entire edge or curve.
- Specify an even distribution or user-defined spacing for the cuts.

Why should I use it?

Use this command to quickly create multiple cuts uniformly spaced along the edge of a plate or stiffener.

Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Edge Cut** 
Menu	**Insert® Steel Features® Edge Cut**

Along Guide Cut

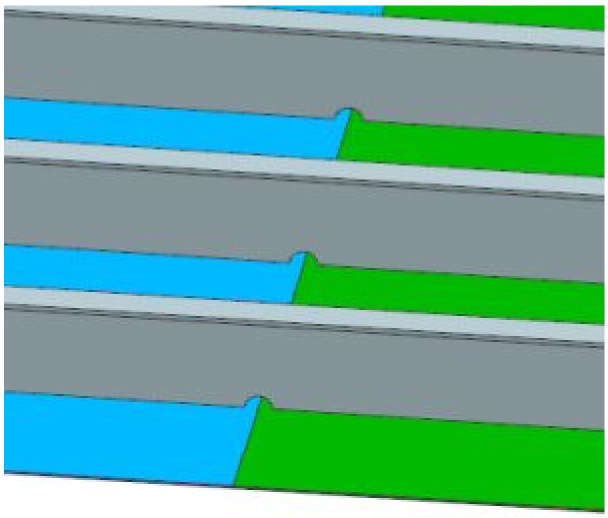
What is it?

The **Along Guide Cut** command replaces the **Along Guide** option previously available in the **Weld Cut** command. Use this command to create multiple cuts on multiple plates and stiffeners where they intersect a selected edge or curve.

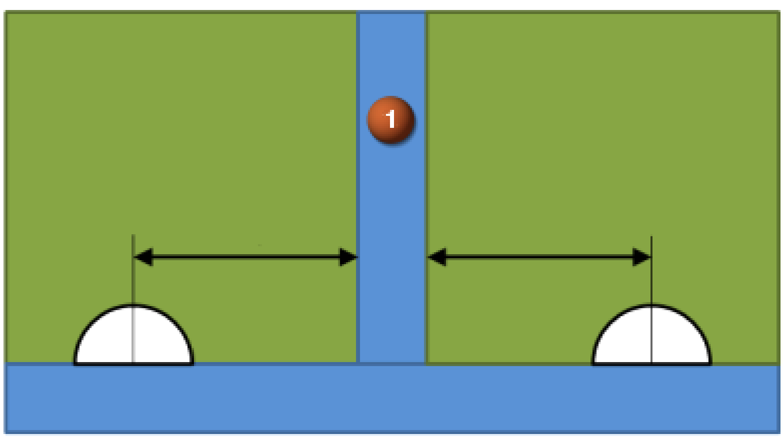
You can:

- Specify a section type from a spreadsheet-driven sketch library. By default, NX provides basic section types.

- Use the **Weld Clearance** placement type to define the direction and orientation of the cut. This is similar to the **Along Guide** option in the previous **Weld Cut** command.



- Use the **Water Stop** placement type to create a pair of symmetric cuts offset from the penetrated plate (1).



Why should I use it?

You can quickly create multiple cuts at the intersections plates and stiffeners.

Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Along Guide Cut**
Menu	**Insert® Steel Features® Along Guide Cut**

Cutout enhancements

What is it?

The **Cutout** command is enhanced to include three new placement types. All of the new placement types let you select a single non-planar placement face or multiple placement faces if the faces are planar and parallel. You can also specify a horizontal reference for the placement of the cross section sketch.

- **Faces + 2 Surfaces** — Lets you select two placement references (solid body, sheet body, face, or datum plane) which may produce nonlinear curves on the placement face. The cutout can be offset from the intersection of the curves.

- **Faces + Between Surfaces + Surface** — Lets you specify two linear placement references used to define a bisector line. The cutout is positioned at intersection of the bisector line and the offset of a third selected placement reference.

- **Faces Centered** — Lets you use the centroid of the selected placement face as the origin of the cutout.

Why should I use it?

The new placement methods give you more control over how the cutout is positioned and allow you to create more than one cutout at a time.

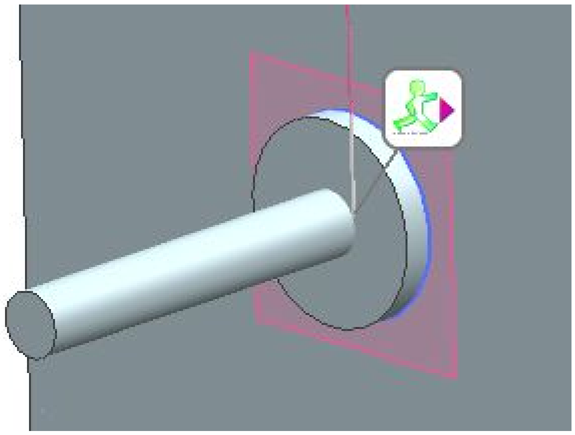
Where do I find it?

Application	Ship Structure Detail Design
Toolbar	**Ship Structure Detail Design® Cutout**
Menu	**Insert® Steel Features® Cutout**

Penetration Management

What is it?

Teamcenter and NX provide the tools to create, track, and respond to requests for the penetration of steel structures in a ship. The penetration may be required for a routing object (pipe, HVAC, electrical), an access panel, or a temporary clearance void.



Placement of pipe through a plate and the object used to create the cutout

Although a penetration request is stored in Teamcenter, you can use NX to:

- Create the penetration request.
- View, manage, and act on the penetration request.
- Place identifiable tags at the location of the requested penetration that are associated to objects in the 3D model.

For information on the penetration request workflow process, see the Issue Manager Guide in the Teamcenter documentation.

Why should I use it?

Use the penetration management tools in Teamcenter and NX to request penetrations and manage the workflow process for getting the appropriate approval. This allows you to manage the penetrations in an effective manner. All of the data associated with the penetrations are stored in the database.

Where do I find it?

Application	Ship Structure Detail Design, Routing Electrical, Routing Mechanical
Toolbar	**Ship Structure Detail Design** or **Routing Mechanical** or **Routing Electrical**® **Penetration Drop-down** list® **Create Request**  and **Verify Penetrations** 
Menu	**Tools**® **Ship Design**® **Penetration Management**® **Create Request** and **Verify Penetrations**

Automated Ship Feature Management

What is it?

Use the **Automated Ship Feature Management** command to delete, hide, or show **Reference Line** or **Marking Line** features in the work part and all of its child components.

Why should I use it?

This command makes it easier to delete, hide, and show ship features that reside in many different parts. You do not have to manually change the work part and select features individually.

Where do I find it?

Application	Ship Structure Basic Design, Ship Structure Detail Design, Ship Structure Manufacturing
Menu	**Tools**® **Ship Design**® **Automated Ship Feature Management**

Display Cutting Side Faces

What is it?

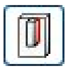
Use the **Display Cutting Side Faces** command to temporarily change the color of cutting side faces on plates in the current session.

Why should I use it?

This lets you easily identify cutting side faces without permanently modifying the faces.

Where do I find it?

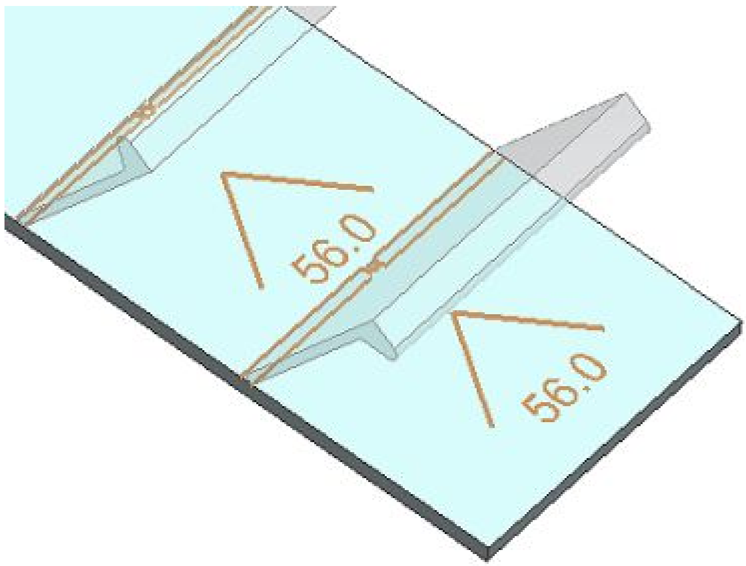
Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Display Cutting Side Faces** 
Menu	**Insert® Manufacturing® Display Cutting Side Faces**



Marking Line enhancement

What is it?

A new **V-type Angle Template** is added to the **Marking Line** customer defaults.



Why should I use it?

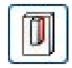
This template can be used instead of the **Y-type Angle Template** to conform to standards at some shipyards.

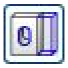
Where do I find it?

Marking Line customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Ship Design® Manufacturing® Marking Line** tab

Create marking lines

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Marking Line** 
Menu	**Insert® Manufacturing® Marking Line**



Excess Material enhancement

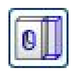
What is it?

The **Excess Material** command is enhanced to automatically delay the modification to the model when the excess material is assigned to the face of a design plate. The modification is delayed until you use the **Plate Preparation** command.

Why should I use it?

This enhancement improves the performance when adding excess material. The ship structure can contain many dependent plates and stiffeners where it is not always desirable to update them immediately.

Where do I find it?

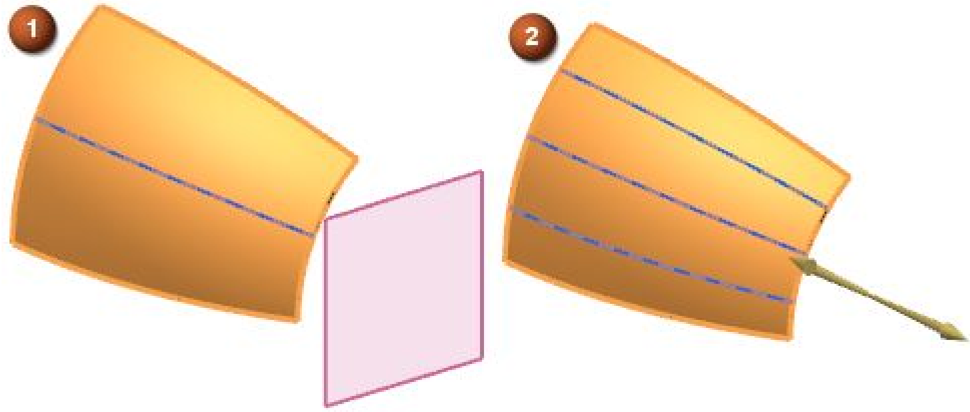
Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Excess Material** 
Menu	**Insert® Manufacturing® Excess Material**

Rolling Line enhancement

What is it?

A new **Stepped Pressure** type is added to the **Rolling Line** command.

Use this option to create a single zero degree isocline curve between parallel planes across the surface of a plate. This differs from the **Pressure** type rolling line which can create many lines on the surface in the same area of the plate.



Stepped Pressure (1) and Pressure (2) type rolling lines

Why should I use it?

This enhancement improves robustness for creating the lines on a plate that guide the rolling machine during the forming process.

Where do I find it?

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing® Rolling Line**
Menu	**Insert® Manufacturing® Rolling Line**

Manufacturing command enhancements

What is it?

The **Ventilation Hole Marking**, **Knuckled Profile**, and **Inverse Bending Lines** commands have been made more robust to produce more consistent and accurate results for different geometry.

Why should I use it?

Use these commands to define the manufacturing shape of the ship structure and to augment that shape with manufacturing information.

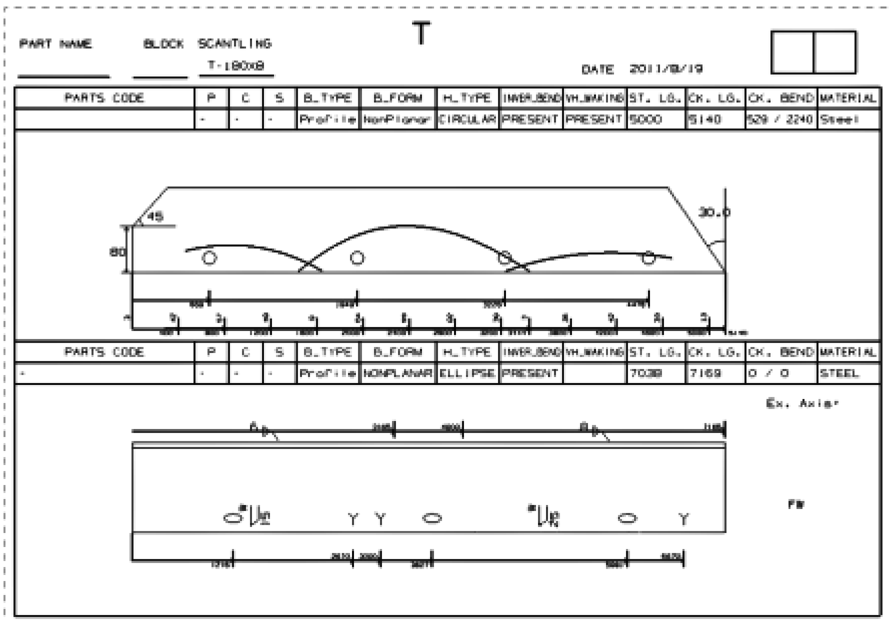
Where do I find it?

Application	Ship Structure Manufacturing
Toolbar	**Ship Structure Manufacturing®** **Ventilation Hole Marking**, **Knuckled Profile**, and **Inverse Bending Lines**
Menu	**Insert®** **Manufacturing®** **Ventilation Hole Marking**, **Knuckled Profile**, and **Inverse Bending Lines**

Profile Sketch

What is it?

Use the **Profile Sketch** command to automatically create drawings of all stiffener profiles in an assembly or workset. The stiffeners are presented as symbolic representations in the drawings and can include end cut and cutout information.



- Stiffeners are grouped by section type.
- Web and toe end cuts are shown in the main view on the drawing. You can define a detail view to show flange end cuts.
- Cutouts can be shown in the main view or in a separate detail view or symbol.

- You can customize the drawings and symbols in the customer defaults. NX provides a sample drawing template configuration and symbol definition files.

Why should I use it?

This command saves you time since you do not have to create the drawings of each stiffener individually.

Where do I find it?

Application	Drafting
Toolbar	**Shipbuilding Drafting® Profile Sketch** 
Menu	**Tools® Shipbuilding® Profile Sketch**

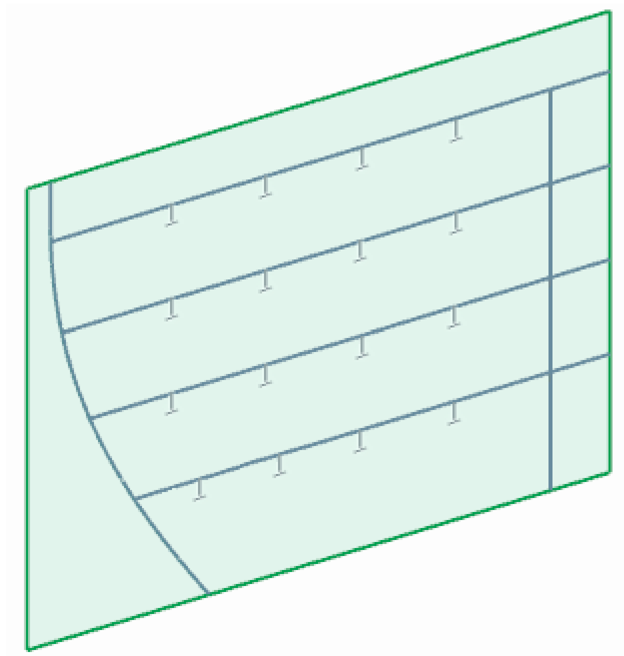
Profile Sketch customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Ship Drafting® Profile Sketch**

Planar Ship Grid enhancement

What is it?

The **Planar Ship Grid** command is enhanced to allow for the selection of basic design features.



You can display the following types of basic design features as curves in a planar ship grid:

- Plate systems (hulls, decks, bulkheads, generic plate systems)
- Stiffener systems
- Edge reinforcement systems
- Standard parts

Why should I use it?

This enhancement saves time by making it easier to select basic design features. If the feature you want to select is not located in the area you are working, you can select the corresponding grid line.

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 Property Filter

What is it?

A new **Ship Property Filter** is added to the **Selection** bar. You can use this option to select structural features that have common properties. For example, you can select features that have the same thickness, section type, material, or end cut.

You have to select the first feature before you can choose the common properties used to select the other features.

Why should I use it?

Use this option to quickly select all features with common properties to edit them.

Where do I find it?

Application	Ship Structure Basic Design, Ship Structure Detail Design, Ship Structure Manufacturing
Toolbar	**Selection® Ship Property Filter**

Detailed Filtering enhancement

What is it?

You can now specify filters for ship objects in the **Detailed Filtering** dialog box.

Why should I use it?

This enhancement makes it easier to select a specific type of ship structure when many different types of objects are displayed close to each other in the graphics window.

Where do I find it?

Application	Ship Structure Basic Design, Ship Structure Detail Design, Ship Structure Manufacturing
Toolbar	**Selection Bar® Detailed Filtering**

Handrail Creator

What is it?

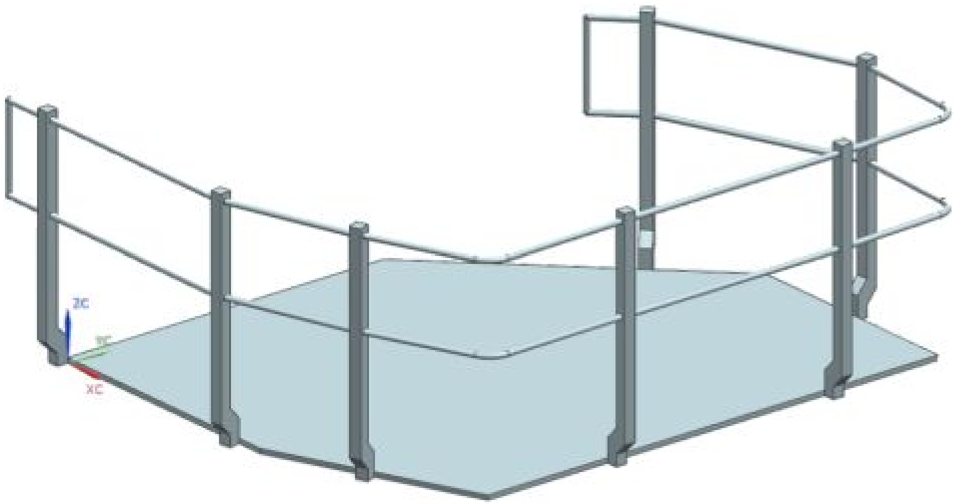
Use this command to build handrails. Each handrail is a subassembly of linear and spline paths, standard parts, and stock.

To use this command, you must create:

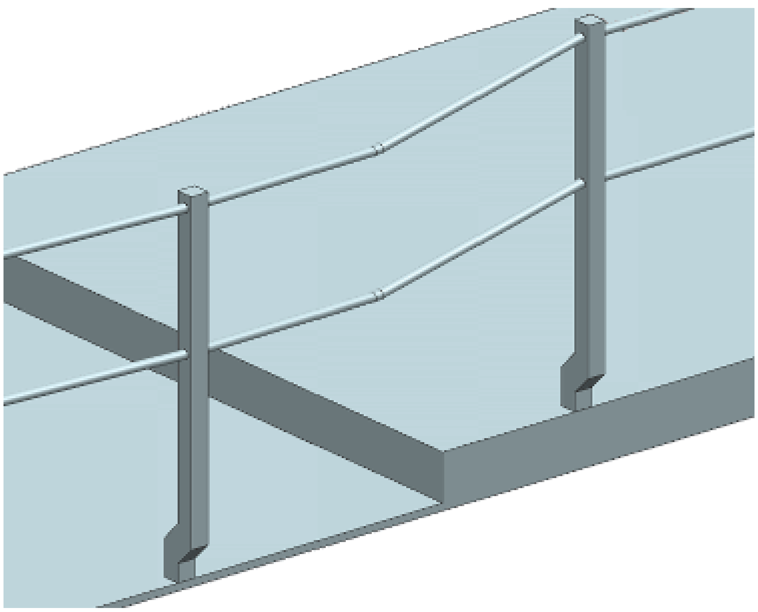
- A Reuse Library of railing stock and handrail posts.
- Part Table files (PTB files) that define the following:
  - o A Start post, which is the post that is placed at the start of the handrail.
  - o An End post, which is the post that is placed at the end of the handrail.
  - o An Intermediate post, which is the post that is placed between the Start and End posts.

Note The PTB files are referenced from the Routing Mechanical Part Library View (PLV) file.

The following example shows handrails on a platform.



The following example shows handrails with bend corners that accommodate the change in elevation.



Why should I use it?

Use this command to easily build handrails on platforms.

Where do I find it?

Application	Routing Mechanical
Toolbar	**Routing Mechanical**® **Tools Drop-down** list® **Handrail Creator**
Menu	**Tools**® **Handrail Creator**

Vehicle Design

All-around Vision

What is it?

Use the **All-around Vision** command to generate the visible or blocked area around the vehicle from the driver's position. This command generates surface or line geometry to represent the blocked or visible area for the driver.

Why should I use it?

You can enhance vehicle design for safe driving by using this command to evaluate the driver's view angles.

Where do I find it?

Application	Modeling
Toolbar	**Vehicle Design® Vision Tools Drop-down** list® **All-around Vision** 
Menu	**Tools® Vehicle Design Automation® Vision Tools® All-around Vision**

Base Data

What is it?

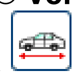
Use the **Base Data** command to configure and edit the base data parameters. This command invokes the **Base Data** dialog box which enables you to configure and edit the following base data parameters:

- Dimensions of the vehicle body such as wheelbase, front and rear body overhang, vehicle length and width, body width, and so on.
- Dimensions and position of wheels such as coordinates of the center point, diameter, static radius, width, track, chamber, and so on.
- Position of the driver such as seat, heel, and pedal reference points, steering wheel center point, and so on.
- Position of passengers such as seat direction and angle, back angle, seat and heel reference point, and so on.
- Define different loading status for the vehicle.

Why should I use it?

Configuring the base data from expressions list is time consuming, tedious, and prone to error. The **Base Data** dialog box makes the configuration of base data expressions simpler. Also, the **Base Data** command automatically updates the related base data geometries based on the changes in the base data expressions.

Where do I find it?

Application	Modeling
Toolbar	**Vehicle Design® Vehicle Architecture Drop-down** list® **Base Data** 
Menu	**Tools® Vehicle Design Automation® Vehicle Architecture® Base Data**

Import and Export Expressions

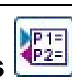
What is it?

You can use the **Import and Export Expressions** command to import/export expressions from/to a spreadsheet.

Why should I use it?

This will improve your efficiency while configuring base data expressions, as you can export expressions and reuse them wherever required.

Where do I find it?

Application	Modeling
Toolbar	**Tools® Import and Export Expressions** 
Menu	**Tools® Vehicle Design Automation® Vehicle Architecture® Import and Export Expressions**

Vehicle Architecture commands

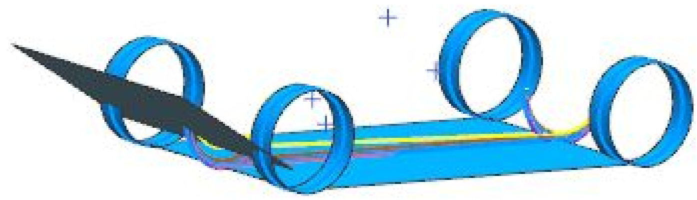
What is it?

New **Vehicle Architecture** commands are available for you to design your vehicles according to the specifications of international standards or according to the customized specific dimensions.

Note All these commands use predefined base data parameters.

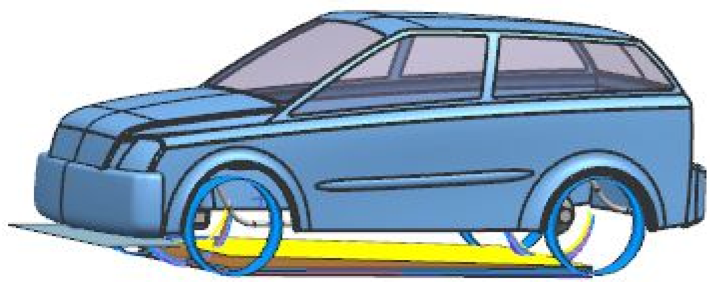
Slope Angle

Use the **Slope Angle** command to create a slope angle surface or curve and to measure slope dimensions for front and rear body objects.



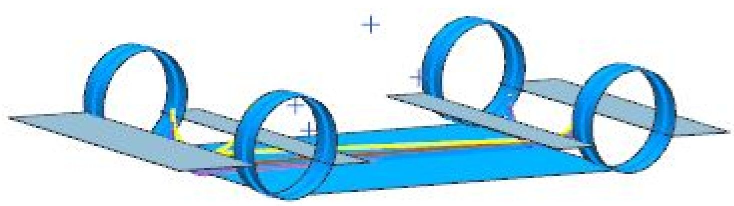
Static Curb

Use the **Static Curb** command to create a static curb surface or curve and to check the clearance between the vehicle geometry and the road surface for front and rear body objects.



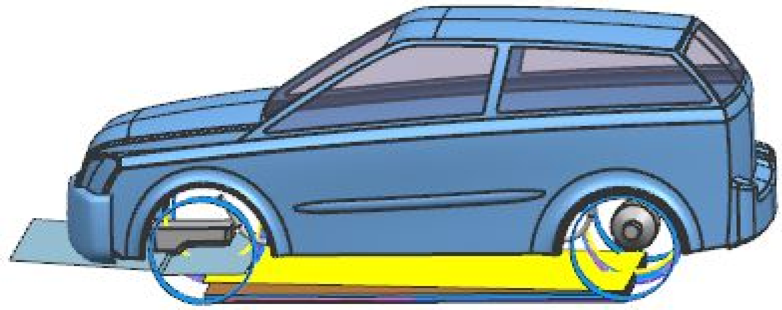
Dynamic Curb

Use the **Dynamic Curb** command to create a dynamic surface or curve and to measure distance between the selected vehicle geometry and the dynamic curb object.



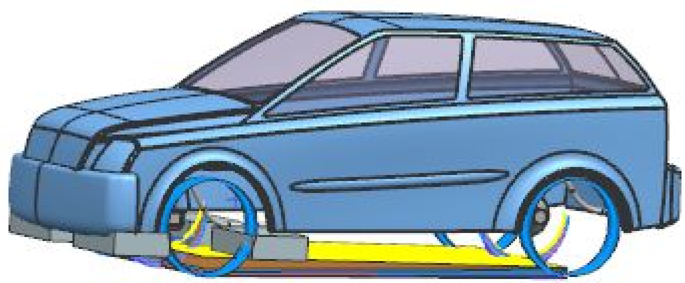
Oil Pan

Use the **Oil Pan** command to create a oil pan surface or curve and to measure distance between the selected vehicle geometry and the oil pan object.



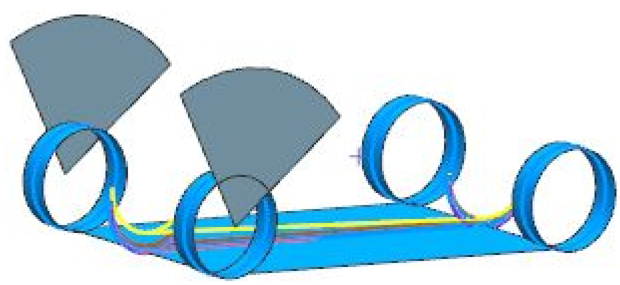
Wheel Fixing

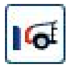
Use the **Wheel Fixing** command to define areas in front of and behind the wheels to help fix the wheels during railway transport.



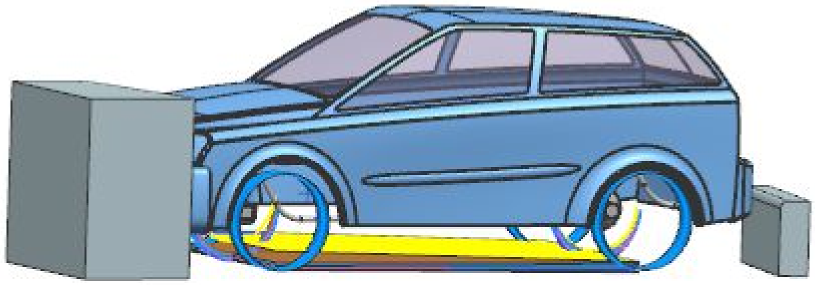
Wheel Covering

Use the **Wheel Covering** command to check wheel covering surface sections and to display intersection curves between vehicle body objects and checked wheel covering surface sections.



 **Crash Barrier**

Use the **Crash Barrier** command to create crash barrier objects which can be used in a real crash test.

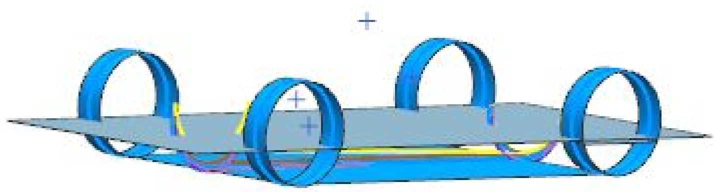


 **Bumper**

Use the **Bumper** command to create bumper pendulum objects which can be used to position bumpers vertically.

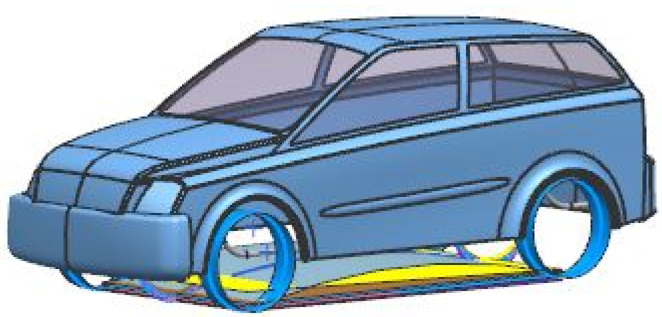
 **Ground Clearance**

Use the **Ground clearance** command to create a ground clearance surface or curve and to measure the distance between the selected vehicle geometry and the ground clearance object.



 **Inner Angle**

Use the **Inner Angle** command to create a inner angle surface or curve and to measure distance between the selected vehicle geometry and the defined inner angle surface.



Where do I find it?

Application	Modeling
Toolbar	**Vehicle Design**® **Vehicle Architecture Drop-down** list
Menu	**Tools**® **Vehicle Design Automation**® **Vehicle Architecture**

Chapter

3 CAM

CAM general

Tool path Verify and Simulate enhancements

What is it?

Enhancements to tool path **Verify** and **Simulate** include the following:

- **Verify** animation speed depends on the part size instead of the display zoom factor.

  The number of IPW display updates for the maximum **Animation Speed** setting is also reduced.

- **Simulate** only creates an IPW for collision pairs which reference the **_WORKPIECE** class when you request the IPW. NX displays an alert message to warn you that collision pairs referencing the **_WORKPIECE** class will be ignored because the IPW is off.

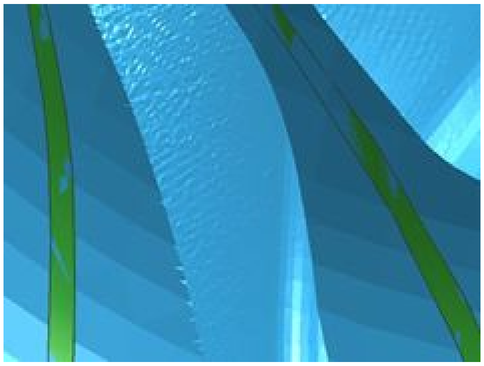
  In NX 8, the IPW was automatically created for any collision pair that referenced the **_WORKPIECE** class.

- **Simulate** fully supports travel to a fixed stop for touch conditions in CSE machine simulation.

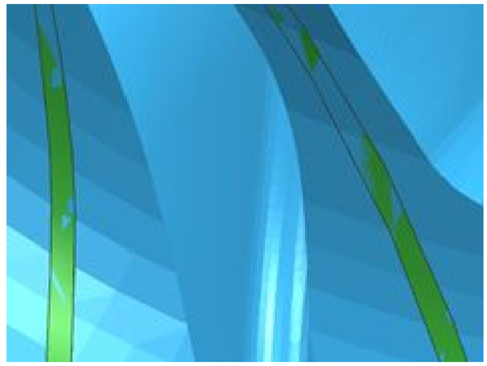
- Faster material removal. There is up to a 50% improvement for 5-axis machining.

- Improved IPW quality.

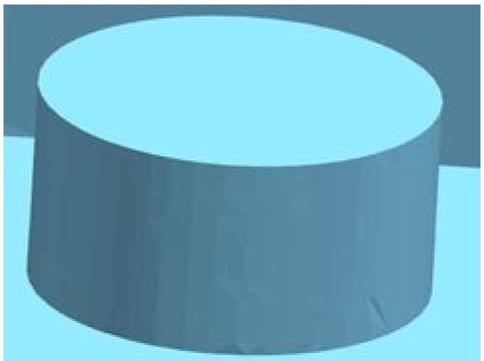
- Improved shading for a smoother surface appearance.



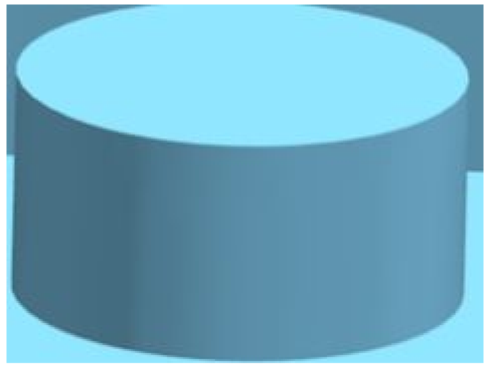
NX 8 IPW



NX 8.5 IPW



NX 8 shading



NX 8.5 shading

Where do I find it?

Application	Manufacturing
Toolbar	**Operations** toolbar® **Verify Tool Path** or **Simulate Machine**
Operation Navigator	Right-click the selected node® **Tool Path**® **Verify** or **Simulate**

Gouge checking enhancements

What is it?

Gouge checking now ignores the following:

- Stock
- Non-uniform custom part offsets
- Negative part offsets that are greater than the tool radius
- Turning tools

NX displays a warning message so that you can choose whether or not to continue.

Why should I use it?

Gouge checking is now clearly defined, and the results are easier to understand.

Where do I find it?

Application	Manufacturing
Toolbar	**Operations** toolbar® **Gouge Check**
Operation Navigator	Right-click the selected node® **Tool Path**® **Gouge Check**

Facet accuracy and NX performance

What is it?

NX uses approximated facet geometry for the In-process workpiece (IPW), gouge detection, and collision detection. The tolerance settings that control the accuracy of facet geometry are now consolidated.

To control IPW accuracy, use the following customer defaults:

- Specify a **Chordal Tolerance** value to control the accuracy required for approximating a curve or circle.

- Choose either **Needle Distance** or **Needle Count** to control the grid used to calculate the IPW.

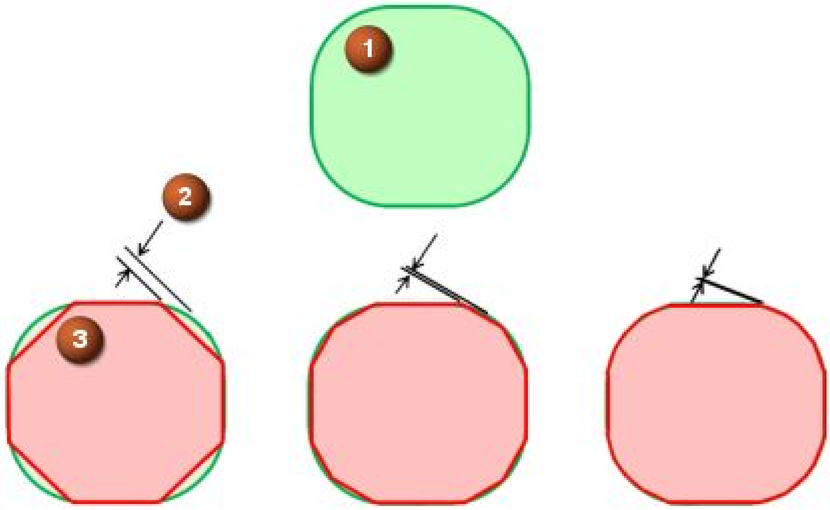
    o **Needle Distance** specifies how far apart to space the grid needles in the X, Y, and Z directions.

    o **Needle Count** specifies the total number of needles in a volume.

  Note The default **Needle Count** values for **Coarse**, **Medium**, and **Fine** should not be modified unless you are directed to do so by technical support.

Chordal tolerance effect on the facets

As the chordal distance decreases, the facet geometry approximation becomes more accurate. The computation time required to create the facet geometry increases with accuracy. You control whether your results are faster or more accurate.



1: Original geometry shape

2: Chordal tolerance

3: Approximated shape for facets

Needle Distance effect on the facets

As the needle distance decreases, the facet geometry approximation becomes more accurate.

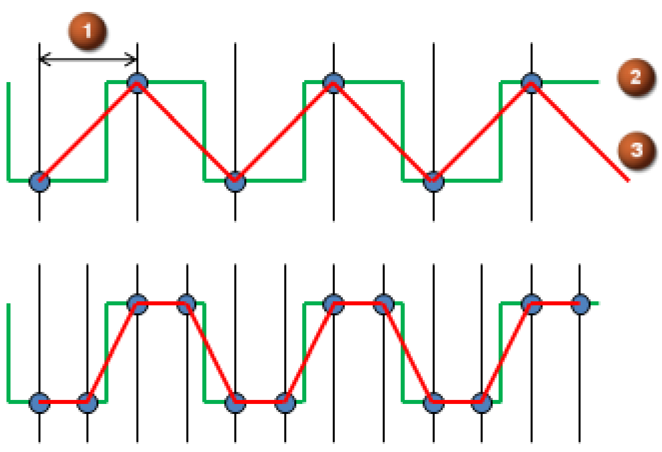
The computation time required to create the facet geometry increases with:

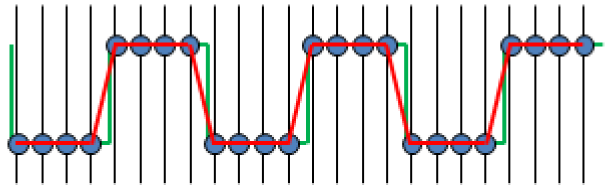
- Part size

  A larger part requires more needles, and more calculations.

- Accuracy

  A shorter needle distance requires more needles and more calculations.





1: Needle distance

2: Original geometry shape

3: Approximated shape for facets

The new **Maximum Needle Count** option restricts the grid size and memory requirements for IPW calculations. Restricting the grid size prevents memory problems that can result from a small needle distance applied to large part geometry.

Why should I use it?

The consolidated options make it easier to adjust the tolerance settings so that they are:

- Large enough to maximize the calculation performance.
- Small enough to create approximated geometry with reasonably sized facets.

Where do I find it?

Customer Defaults

Application	Manufacturing
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Customer Defaults** dialog box® **Manufacturing**® **Simulation & Visualization**: (**Chordal Tolerance** and needle control for IPW facets) **IPW** tab (**Chordal** tolerance for gouge checking) **Gouge** tab (**Faceting Tolerance for Collision Detection**) **ISV** tab

(**Chordal Tolerance** for machine simulation)

Toolbar	**Operations** toolbar® **Simulate Machine**
Location in dialog box	**Simulation Control Panel** dialog box® **Simulation Settings** group® **Options** ® **Simulation Options** dialog box® **Other Options** group

Tool export reports

What is it?

When you successfully export a tool to the ASCII tool library or a Teamcenter library, NX generates a report of the result of the export, displays a simplified version of the report in a message, and writes a detailed version of the report to the log file.

The summary report has information about the name of the library where the tool and holder were created or updated, and whether tool kinematics for the Machine Tool Builder application were created. The detailed report contains the same information, and it also shows the path to the library if you are running native NX; whether tracking point data was written to the part attributes; and where the tool part file was saved in native NX, or, whether the tool part file was saved in Teamcenter.

Where do I find it?

Application	Manufacturing
Operation Navigator	Edit a tool
Location in dialog box	**Library** group, **Export Tool to Library**

Tool pocket capacity

What is it?

In the Machine Tool Builder application, when you define a pocket in a machine tool kinematic structure, you can now enter a tool capacity for the pocket. The default capacity value is 1. When you add the machine tool to your CAM setup in NX, the capacity value is copied to the pocket parameters in the **Operation Navigator**.

NX does not list full pockets as possible parents in the **Create Tool** dialog box. You also cannot drag tools into full pockets.

Why should I use it?

You can ensure that the number of tools assigned to a pocket is valid.

Where do I find it?

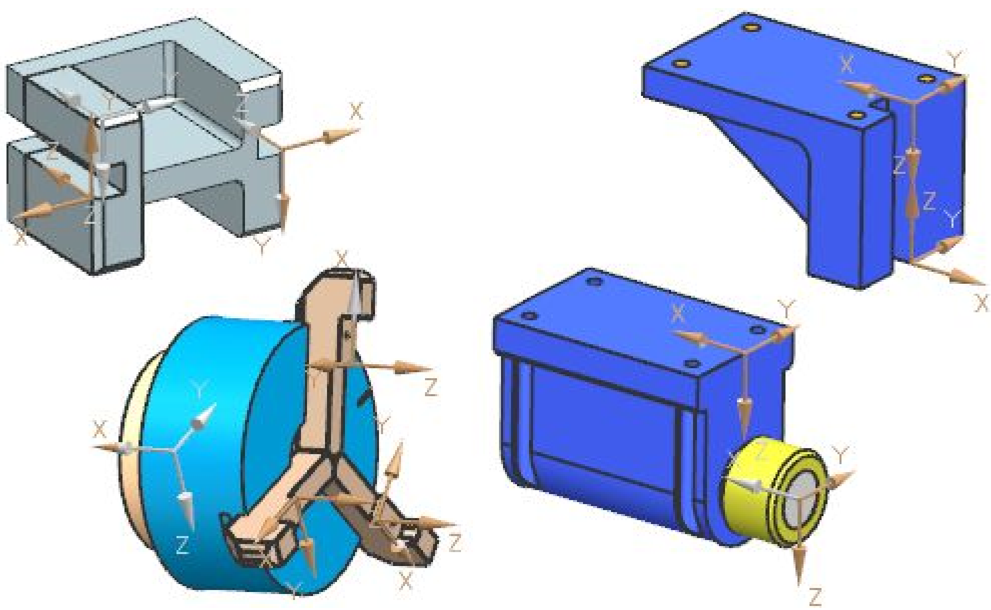
Application	Manufacturing
Operation Navigator	Right-click a **Pocket** node® **Edit**
Location in **Pocket** dialog box	**Capacity** group ® **Number of tools**
Machine Tool Navigator	Right-click a **Pocket** node® **Edit**® **Machine Component**
Location in **Edit Machine Component** dialog box	**Holder Details** group® under **Capacity**® **Number of Tools**

Retrieve device from library

What is it?

You can retrieve devices from the new device library. If you select a pocket before you click **Create Tool**, the device is automatically assigned to the selected pocket.

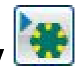
The following figure shows some of the lathe devices that are available in the supplied library, and their junctions.



Why should I use it?

You need not search for a newly retrieved device and assign it to a pocket.

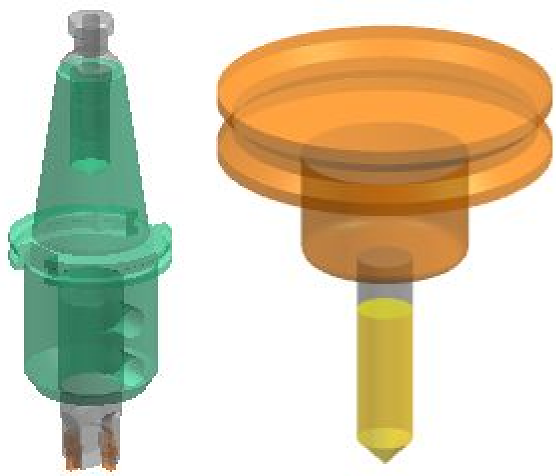
Where do I find it?

Application	Manufacturing
Prerequisite	To automatically assign a pocket, select the target pocket in the **Operation Navigator** before you retrieve your device.
Toolbar	**Insert**® **Create Tool** 
Menu	**Insert**® **Tool**
Location in dialog box	**Library** group® **Retrieve Devices from Library** 

Tool holder display

What is it?

When you select or edit a tool, NX now displays the defined holder or device to which the tool is mounted.



Why should I use it?

You can verify if the defined holder is correct and appropriate for applications such as simulation with collision checking.

Where do I find it?

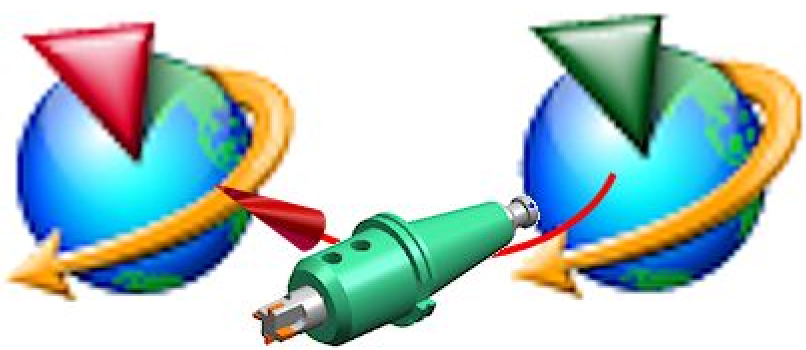
Application	Manufacturing
Toolbar	**Insert® Create Tool** 
Operation Navigator	Select a tool Rick-click a tool® **Edit** Double-click a tool

Retrieve tools from library enhancement

What is it?

In native NX, when you retrieve cutting tools from a library, you can now access tool components and tool assemblies from a Teamcenter Manufacturing Resource Library (MRL). Previously, only the ASCII library in the *MACH* folder structure was available in native NX.

Your administrator must set up MRL Connect for NX on a four-tier Teamcenter server.



Tools flow in one direction, from Teamcenter to native NX

Information for administrators

To set up MRL Connect for NX, you must do the following:

1. On the Teamcenter server, from the Teamcenter installation disk, from the *advanced_installations\resource_management* folder, run the *setup.exe* program to install the MRL in your database.

2. On a computer with NX installed, again run the MRL *setup.exe* program, and select the **Configure NX Library** option.

   These files are copied into a specified *MACH* structure:

   - *...mach\resource\library\tool\inclass\dbc_mrl_tooling_library_tlas.tcl*

- *...mach\resource\library\tool\inclass\dbc_mrl_tooling_library_tlas_en.def*
- *...mach\resource\library\tool\inclass\dbc_mrl_tooling_library_tlas_de.def*
- *...mach\resource\ug_library\dbc_mrl_general.tcl*

3. After the installation, copy these files to every instance of *MACH* or *RESOURCE* that NX users will be using when they access the MRL.

   The *.def* files which control the user interface to the MRL are currently released for English (en) and German (de). Other languages may be added.

4. On each computer that has an NX environment, define the following environment variables to suit your database:

   - FMS_HOME
   - JAVA_HOME
   - UGII_UGMGR_COMMUNICATION — typically, this is HTTP.
   - UGII_UGMGR_HTTP_URL — this is the URL to access four-tier services on Teamcenter.

5. Make certain that four tier services are started and available to the NX users on your network.

Where do I find it?

Application	Manufacturing
Prerequisite	In Preferences® Manufacturing, choose the *cam_native_rm_library.dat* configuration file, or any CAM configuration file that your administrator has configured MRL Connect for NX.
Toolbar	**Insert**® **Create Tool**
Menu	**Insert**® **Tool**, or edit an existing tool
Location in dialog box	**Library** group® **Retrieve Tools from Library** 

Show Balloon Tooltips in Dialog Options

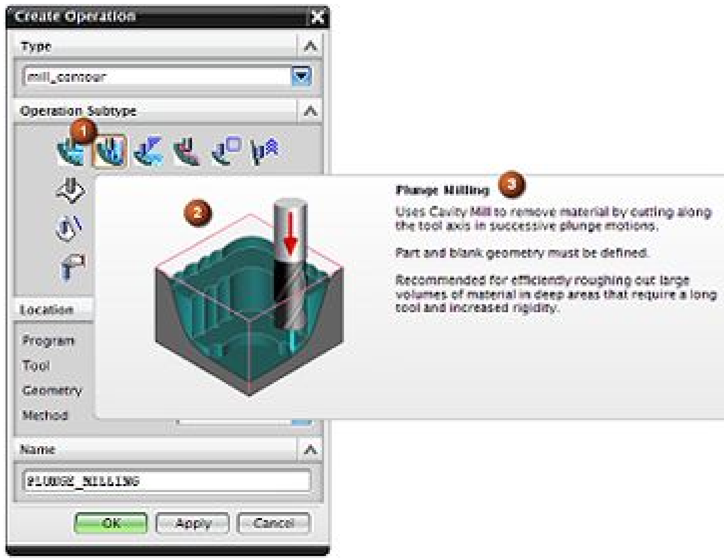
What is it?

When you pause your cursor over an option name in the **Create Operation** dialog box, NX displays a graphic that illustrates the option.

1. Dialog box **Operation Subtype**.

2. Picture illustrating what the operation does.

3. Title and description of what the operation does, geometry requirements, and recommended usage.



These graphics are displayed when:

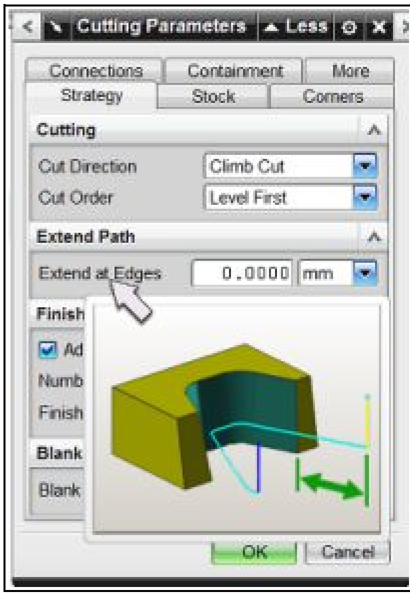
The **Show Balloon Tooltips** check box on the **Dialog Options** dialog box is selected (Right-click in toolbar® **Customize**® **Options** tab).

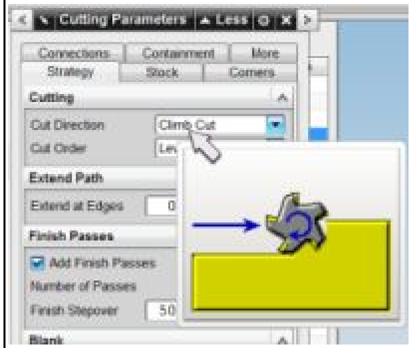
The **Show Picture Tooltips** check box on the **Create Object Dialog Boxes** dialog box is selected (**File**® **Utilities**® **Customer Defaults**® **User Interface**® **Dialog Boxes** tab).

Tooltips in the Cutting Parameters dialog box

What is it?

When you pause your cursor over an option name in the **Cutting Parameters** dialog box, NX displays a graphic that illustrates the option.





These graphics are displayed when:

The Show Balloon Tooltips in Dialog Options check box is selected.

The Show Picture Tooltips in Create Object Dialog Boxes customer default is selected.

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

Where do I find it?

Application	Manufacturing
Toolbar	Right–click in toolbar® **Customize**
Location in dialog box	**Options** tab® **Tooltips** group® **Show Balloon Tooltips in Dialog Options**.

Milling enhancements

Volume based 2.5D milling

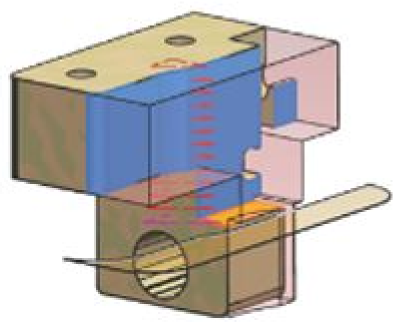
Volume based 2.5D milling

What is it?

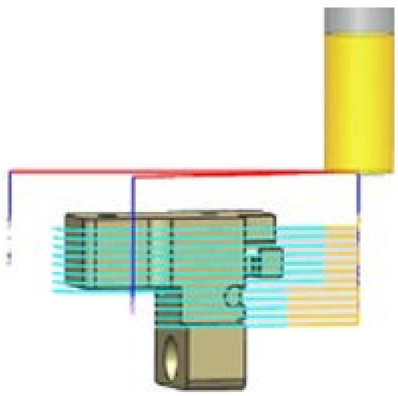
Use the new **Floor Wall Milling**  operation subtype to efficiently machine prismatic parts and features. The **Floor Wall Milling** operation replaces Face Milling operations, and provides enhanced functionality.

You can do the following:

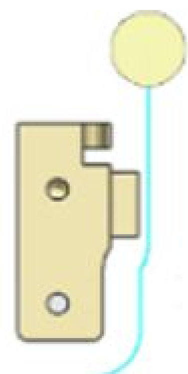
- Simultaneously machine floors, walls, and floor and wall combinations.
- Machine walls that are not bounded by floor geometry, such as the outside periphery of a part.
- Control the blank.
- Contain cut regions using either the **Floors** or **Walls** option.
- Control the cut region shape using the **Extend Walls** option.
- Machine tapered walls using the **Exact Positioning** option.
- Preview the blank and the cut levels.



Part and IPW showing the cut region and cut levels



Tool path cutting along the wall down to the floor



Top view of tool path

Legacy Face Milling operations

NX will migrate legacy Face Milling operations to the appropriate operation type.

NX 8 operations	NX 8.5 operations
FACE_MILLING_AREA, using the **Follow Part**, **Follow Periphery**, **Trochoidal**, **Zig**, **Zig Zag**, **Zig with Contour** cut patterns	**Floor Wall Milling**
FACE_MILLING_AREA, using the **Mixed** cut pattern	**Face Milling Manual**
FACE_MILLING_AREA, using the **Profile** cut pattern	**Face Milling using Cut Area**, using the **Profile** cut pattern
FACE_MILLING, using boundaries	**Face Milling with Boundaries**

Why should I use it?

Use the **Floor Wall Milling** operation to increase efficiency and reduce programming time of 2.5 D prismatic parts and features.

Where do I find it?

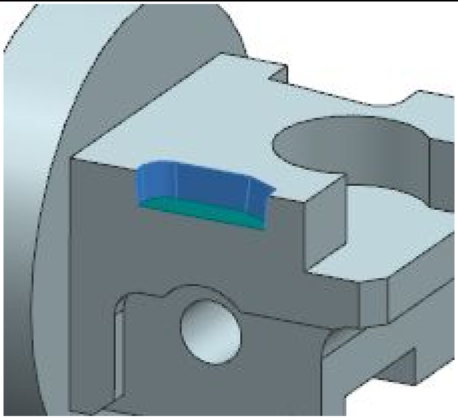
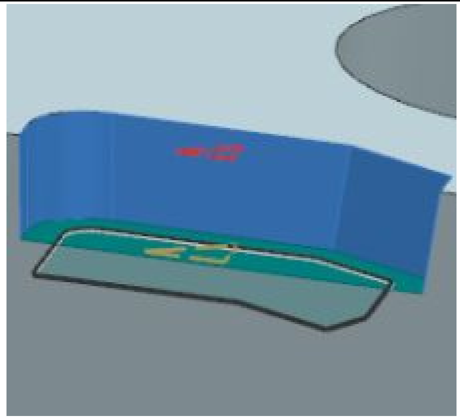
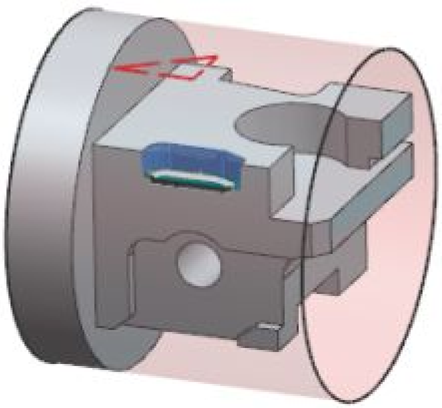
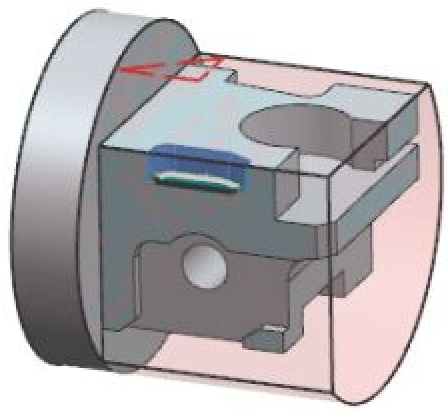
Application	Manufacturing
Toolbar	**Insert® Create Operation**
Menu	**Insert® Operation**
Location in dialog box	**Type** list® **cam_test_new® Operation Subtype** group® **FLOOR_WALL_MILLING**

Floor Wall Milling blank control

What is it?

You specify one of the following options to define the blank input for a **Floor Wall Milling** operation:

- **Blank Geometry** uses the blank defined in the **Workpiece** geometry group.
- **3D IPW** uses 3D IPW geometry from prior operations in the same geometry group.
- **Thickness** lets you specify a blank thickness value to apply to the floors and walls. You can define the blank when it is not defined in the part.

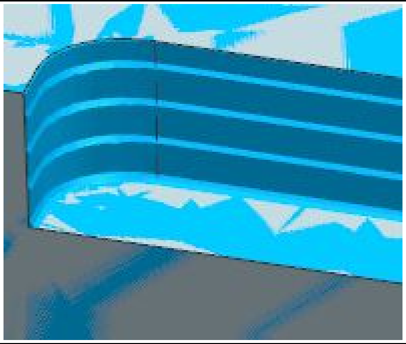
 Cut area	 Preview of **Blank** = **Thickness** Using **Wall Blank Thickness**
 Preview of **Blank** = **Blank Geometry**	 Preview of **Blank** = **3D IPW**

Where do I find it?

Application	Manufacturing
Location in dialog box	**Floor Wall Milling** dialog box® **Path Settings** group® **Cutting Parameters** ® **Cutting Parameters** dialog box® **Containment** tab® **Blank** group

Floor Wall Milling cut region containment using Floors or Walls

Use the **Floors** option to project all cut levels along the tool axis. Use the **Walls** option to efficiently machine regions with tapered or contoured walls.

	
4 cut levels **Cut Region Containment** = **Floors**	4 cut levels **Cut Region Containment** = **Walls**

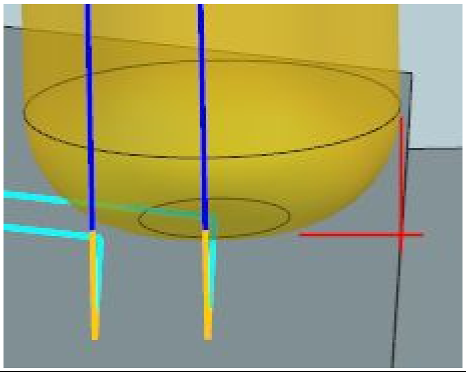
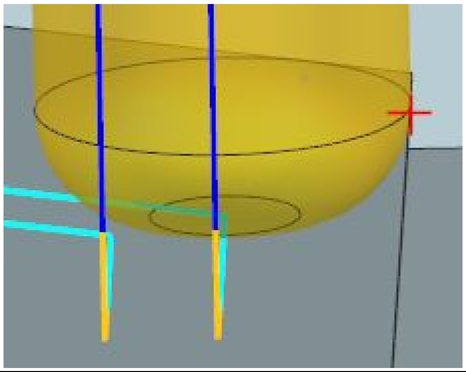
Where do I find it?

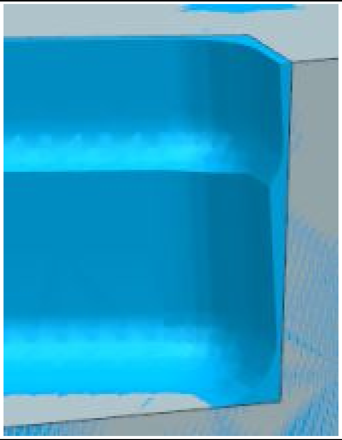
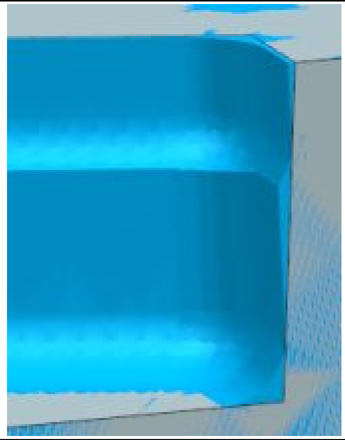
Application	Manufacturing
Location in dialog box	**Floor Wall Milling** dialog box® **Path Settings** group or **Floor Wall Milling** dialog box® **Path Settings** group® **Cutting Parameters** ® **Containment** tab® **Cut Regions** group

Floor Wall Milling Exact Positioning

What is it?

If the tool for your operation is an end mill with a corner radius, use the **Exact Positioning** option to control how the cutter is positioned to tapered walls.

	
☐ **Exact Positioning**	☑ **Exact Positioning**

Where do I find it?

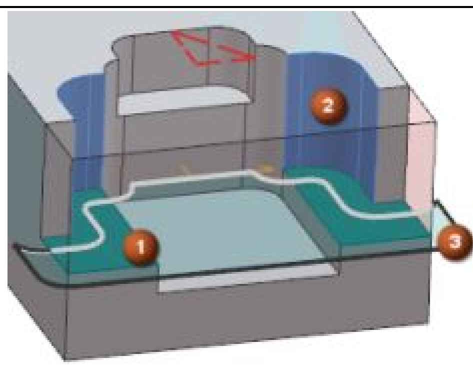
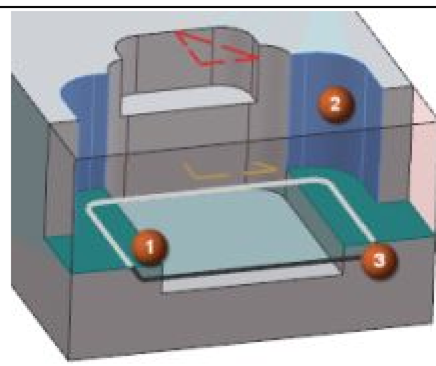
Application	Manufacturing
Location in dialog box	**Floor Wall Milling** dialog box® **Path Settings** group or **Floor Wall Milling** dialog box® **Path Settings** group® **Cutting Parameters** ® **Cutting Parameters** dialog box® **Containment** tab® **Cut Regions** group

Floor Wall Milling Extend Walls

What is it?

Use the **Extend Walls** option to control the cut region shape when the floor is not completely bounded by walls.

In the following example, 1= Cut area floor, 2= Wall geometry, and 3= Cut region preview.

	
☐ **Extend Walls**	☑ **Extend Walls**

Where do I find it?

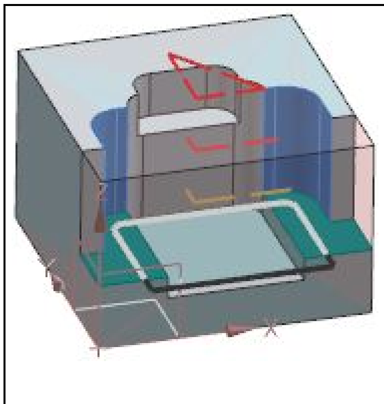
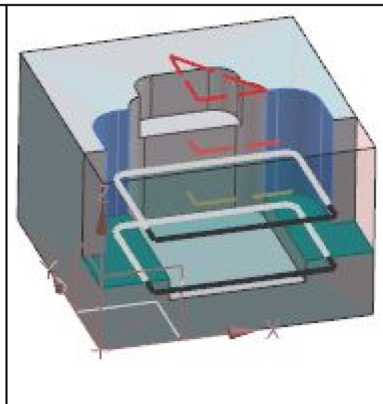
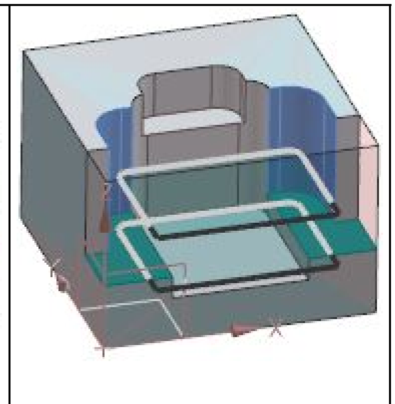
Application	Manufacturing
Location in dialog box	**Floor Wall Milling** operation dialog box® **Path Settings** group® **Cutting Parameters** ® **Cutting Parameters** dialog box® **Containment** tab® **Cut Regions** group

Floor Wall Milling Preview

What is it?

In **Floor Wall Milling** operations, you can:

- Continuously display the cut levels and the cut regions for critical levels by selecting the **Preview** check box.
- Display the cut regions for all cut levels using the **Display** option.

		
Preview	**Preview** **Display**	**Preview** **Display**

Where do I find it?

Application	Manufacturing
Location in dialog box	**Floor Wall Milling** dialog box® **Preview** group

Flow Cut Enhancements

What is it?

The **Flow Cut** drive method was primarily focused on finishing conditions, and has now been enhanced to semi-finish areas on the part with uncut material, such as corners, deep pockets, or slots. The **Flow Cut Reference**

Tool operation references a larger tool to determine the areas on the part where material remains, then cuts with a smaller tool to remove the material.

The following new **Zlevel** cut patterns are supported for the **Flow Cut** drive method:

- **Zlevel Zig**
- **Zlevel Zig Zag**
- **Zlevel Zig Zag with Lifts**

The new **Zlevel** cut patterns let you do the following:

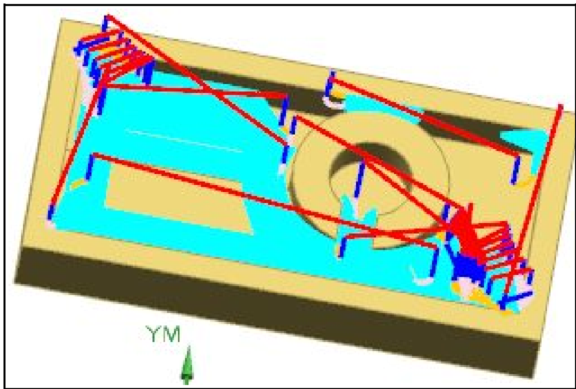
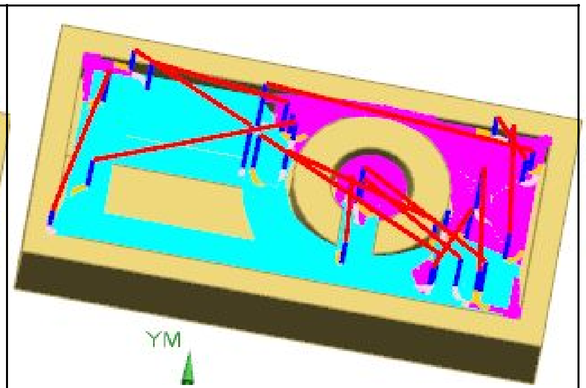
- Machine slots and pockets with steep walls.
- Machine near vertical corners where **Zlevel** gives the best results.

  The **Zlevel** cut pattern keeps the tool on a plane in steep areas to prevent tool motions that plunge into material.
- Machine pockets where there is more material remaining on the floor than in the steep corners.
- Remove uncut material that would otherwise remain when the reference tool for a **Flow Cut Reference Tool** operation is too large to fit inside a pocket or slot.

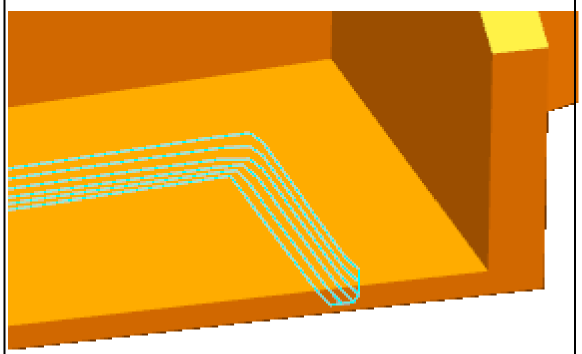
You specify the cut patterns for steep and non-steep areas separately, and can improve machining results for semi-finishing operations by using one of the Flowcut patterns in shallow areas (non-steep) and one of the Zlevel cut patterns in steep areas.

The **Flow Cut** operation creates smooth transitions between the Zlevel and Flow Cut cut patterns as well as steep and non-steep regions.

Remove material where the reference tool is too large to fit

	
Non-steep Cut Pattern = **Zig Zag** **Steep Cut Pattern** = **Crosscut Zig**	**Non-steep Cut Pattern** = **Zig Zag** **Steep Cut Pattern** = **Zlevel Zig Zag**

Zlevel for steep areas

	
Steep Cut Pattern = **Zlevel Zig Zag**	**Non-steep Cut Pattern** = **Zig Zag**

Where do I find it?

Application	Manufacturing
Prerequisite	Must follow a previous operation that used a larger tool.
Location in dialog box	**Drive Method** group® **Flow Cut**® **Flow Cut Drive Method** dialog box

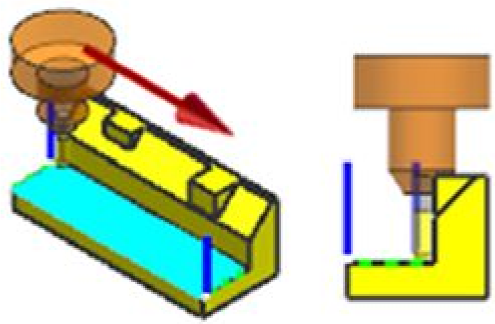
Tool Axis Tilt enhancements

What is it?

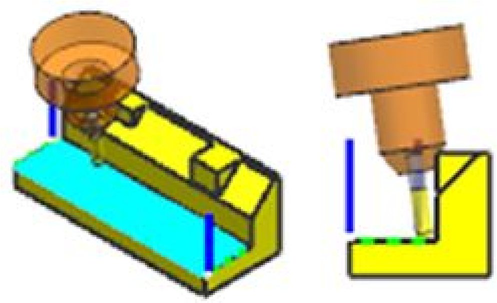
The **Tool Path Tilt** dialog box is now renamed **Tilt Tool Axis**. New options are added to the dialog box to give you additional control over the tilting behavior of the tool axis.

Default tool behavior

- The tool begins cutting in the original fixed tool axis orientation.



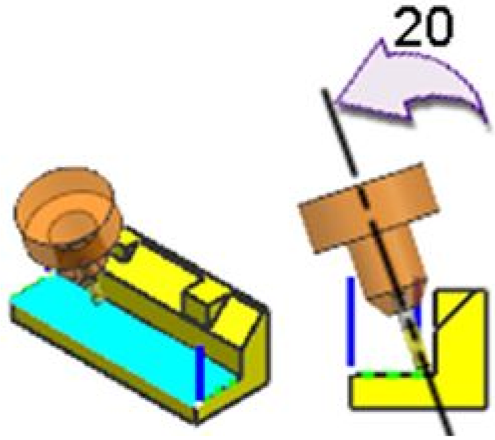
- As NX encounters a potential collision, it tilts the tool minimally to avoid the collision and then returns the tool to the original tool axis orientation. NX repeats these tool motions when it encounters the next collision.



New Options:

- Use the new **Preferred Tilt from Fixed Axis** option to specify a tilt angle value. NX tilts the tool by this value as it attempts to avoid the first collision, and then it returns the tool to its original tool axis orientation. It repeats these motions when it encounters the next collision.

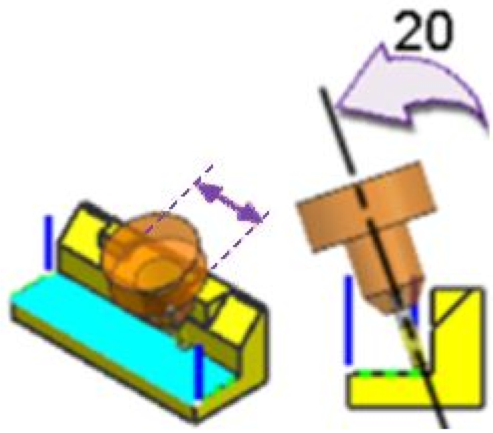
  In this example, the **Preferred Tilt from Fixed Axis** = **20**.



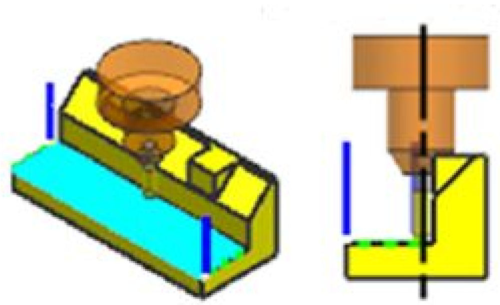
Note If the value you enter is not appropriate to avoid a collision, NX will allocate a different gouge free tool axis orientation.

- Select the **Maintain Current Tilt between Collisions** check box to keep the tool from tilting back to its origin tool axis orientation between collisions. This option reduces unnecessary rotary tool axis movement.

  If you select this check box, you can use the **Max Distance Between** option to specify the distance within which the tool maintains the current tool axis tilt angle between collisions.

  If the **Max Distance Between** value is less than the actual distance between collisions, the tool returns to the original tool axis orientation after the first collision.



Why should I use it?

You can use the new options to reduce rotary axis changes by controlling whether the tool axis returns to the default tilt between features.

Where do I find it?

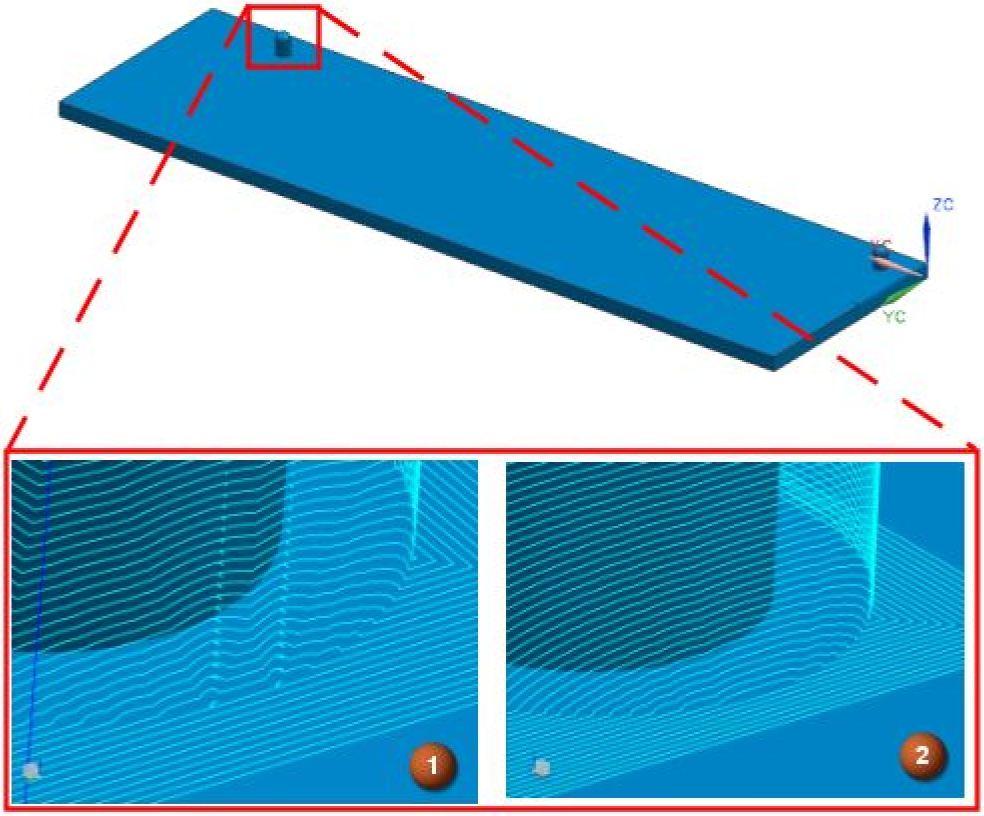
Application	Manufacturing
Prerequisite	• The operation must be a Surface Contouring or Zlevel operation. • The tool must have a ball shape (**BALL_MILL**, **SPERICAL_MILL**, or **MILL**) where the lower radius of the tool = ½ of the diameter of the tool. • The tool must have a shank and or a holder defined. • Before tool path generation, the **Check Tool and Holder check** box must not be selected in the **Cutting Parameters** dialog box® **Containment** tab® **Collision Checking** group.
Operation Navigator	Right click an operation® **Tool Path**® **Tilt Tool Axis**

Surface Contouring processor enhancements

What is it?

The fixed axis surface contouring processor has been enhanced to improve operations on large parts that contain multiple small cut areas or trim boundaries.

The processor calculates each small tool path locally to improve the accuracy of the tool path, the calculation speed at which it is created, and the quality of the surface finish.



1	Previous versions of NX
2	NX 8.5

Cavity Mill processor enhancements

What is it?

The Cavity Mill processor has been enhanced to improve the accuracy of the tool path, the calculation speed at which it is created, and the quality of the surface finish.

This behavior applies to following Milling subtypes:

- Cavity Mill
- Zlevel
- Zlevel 5–Axis
- Fixed Axis Surface Contour Area
- Plunge Milling
- Flowcut

Variable-axis surface contouring enhancments

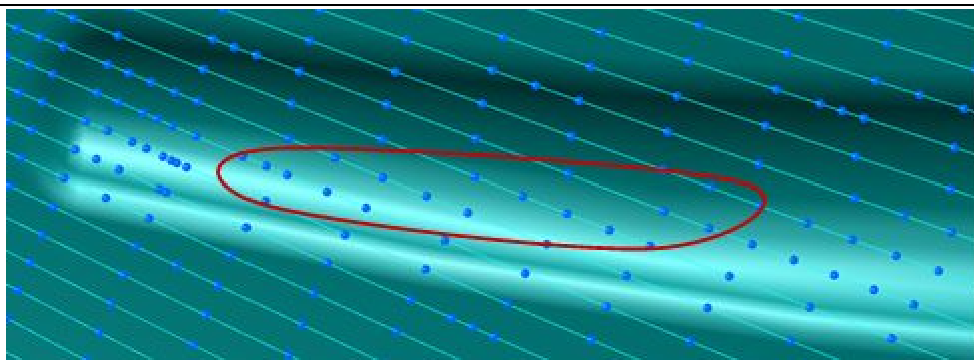
What is it?

Enhancements to variable-axis surface contouring operations do the following:

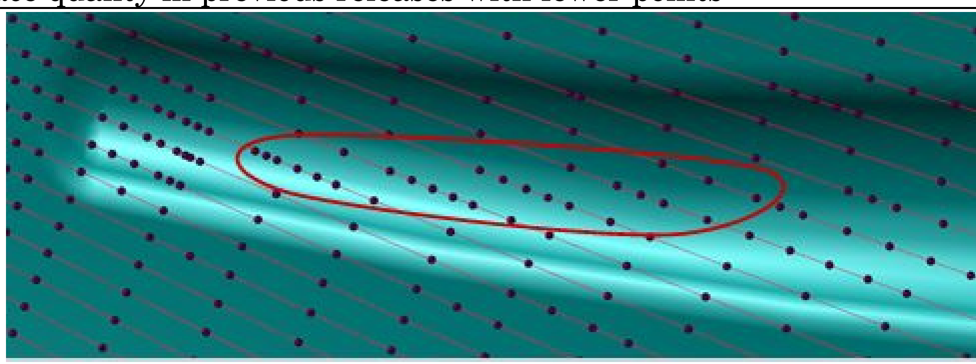
- Improve the surface quality when cutting across small features.

  NX adds points along the path in high curvature regions.

- Reduce tool path generation time.



Surface quality in previous releases with fewer points



NX 8.5 behavior

Where do I find it?

Application	Manufacturing
Toolbar	**Insert® Create Operation** 
Menu	**Insert® Operation**
Location in dialog box	**Create operation** dialog box® **Type® mill_multi_axis® Operation Subtype® Variable Contour**

Zig pattern enhancements

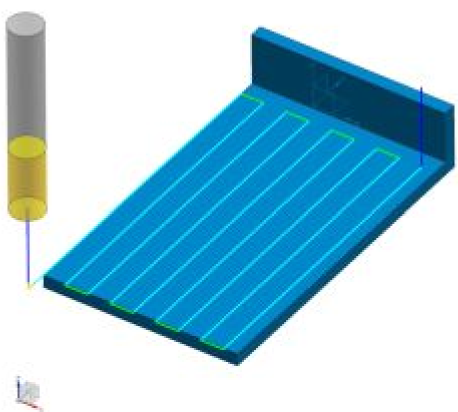
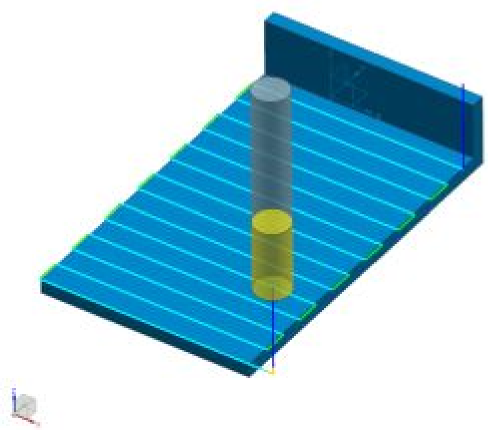
What is it?

The Zig and Zig Zag patterns in a Floor Wall Milling operation are enhanced to give you improved cutting efficiency and greater control over the tool path.

Automatic cut angle option

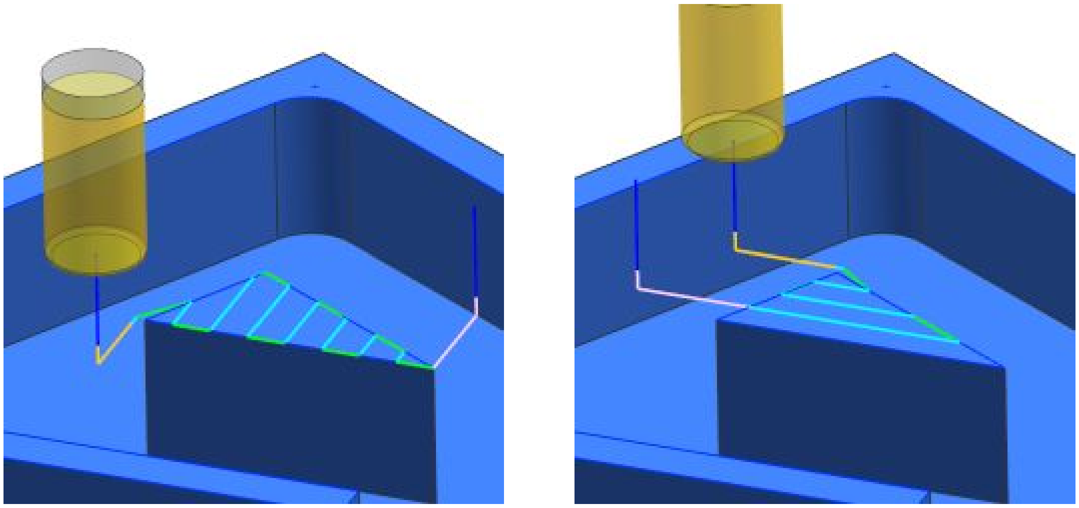
The **Automatic** cut angle option is enhanced in the following ways:

- If the milled faces are bound by one or more walls, NX orients the milling direction to follow the direction of the wall.

Previous Releases	NX 8.5
	

- If the faces are not bound by walls, NX minimizes the number of passes and makes each pass as long as possible.

Previous Releases	NX 8.5



Stepover distance and pass control

You can:

- Precisely control the stepover distance.

  Use the new **Exact** option to specify the stepover distance.

- Precisely control the number of passes.

  Use the new **Passes** option to specify the number of passes.

- Control the spacing, but allow the stepover distance to vary slightly with the cut area size.

  Use the new **Variable Average** option to specify the minimum and maximum distance values. NX analyzes the first pass and the next wall, and then creates equal spaces in between them.

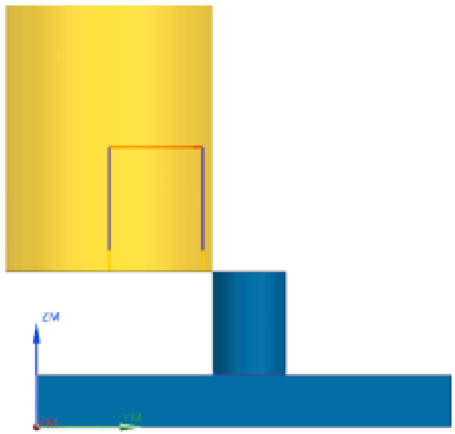
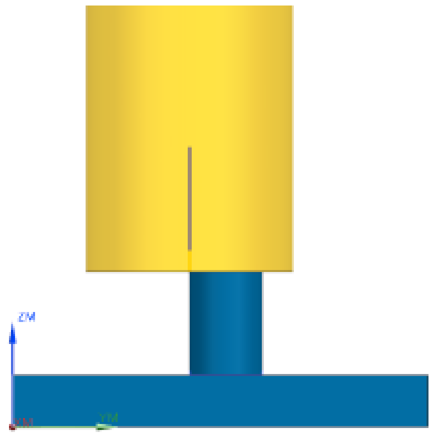
Cut Regions

You now have better control over the cutter path by specifying the following new input values in the **Cutting Parameters** dialog box.

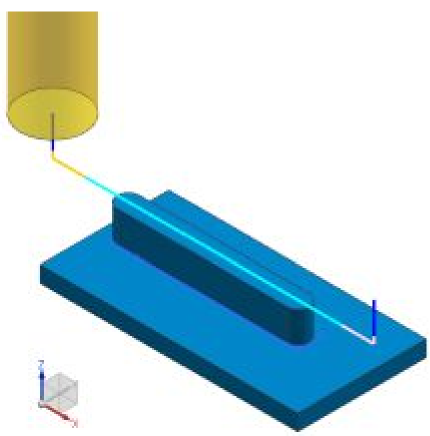
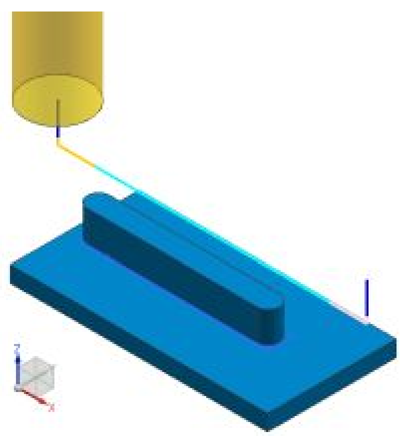
- **First Pass Overhang** specifies the percentage of the cutter diameter or the distance by which the tool hangs over geometry on the first pass of your cutter path. This option replaces the **Blank Overhang** option.

- **Tool Run On** specifies the percentage of the tool diameter or distance by which to extend the cutting motion before the tool reaches the cut region.

- **Tool Run Off** specifies the percentage of the tool diameter or distance by which to extend or trim cutting motion relative to the cut region.

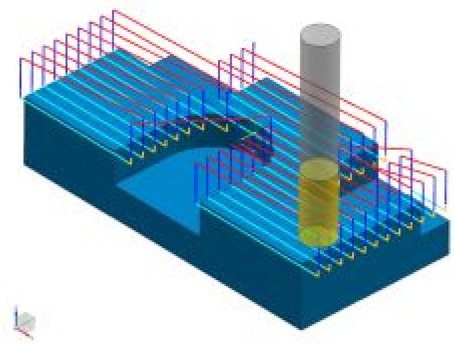
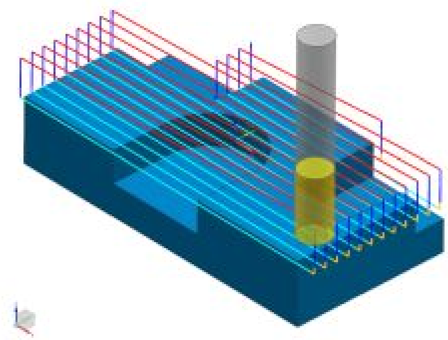
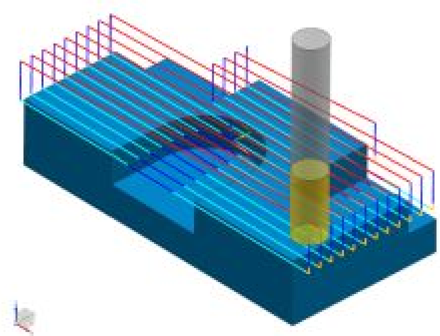
First Pass Overhang, **Run On**, or **Run Off** at 100% of tool diameter.	**First Pass Overhang**. **Run On**, or **Run Off** at 50% of tool diameter.
	

- **Single Pass Offset** lets you select an offset distance for the path only when single path is selected.

Single Pass with no offset. Path is centered on geometry.	**Single Pass** with 50% of tool offset parallel to path and only over the part area.
	

Across voids enhancement

The Across Voids options let you handle open voids much like closed voids in previous versions. You have 3 options to control the cutter behavior over these regions:

Follow allows the cutter to follow the exact geometry with lifts	
Cut will follow the geometry but will create a continuous path over the void.	
Traverse will follow the geometry but will traverse between engagements of part geometry to be machined.	

Where do I find it?

Application	Manufacturing
Prerequisite	Floor Wall Milling operation
Location in dialog box	[Floor Wall Milling operation dialog box]® **Path Settings** group® **Cutting Parameters** →**Strategy** tab® **Cutting** group (Automatic cut angle) **Cutting Parameters** ® **Strategy** tab® **Cutting** group (Tool Overhang) **Cutting Parameters** ® **Containment** tab® **Cut Regions** group (Pass Control) [Floor Wall Milling operation dialog box]® **Path Settings** group® **Stepover** list (Across Voids) **Cutting Parameters** ® **Connections** tab® **Across Voids** group® **Motion Type** list

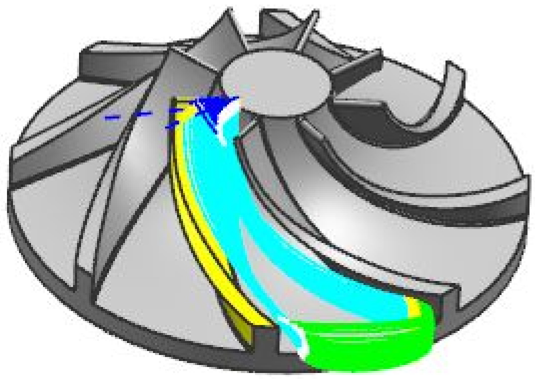
Turbomachinery — Multi-Blade milling enhancements

Multi-Blade milling enhancements

What is it?

Enhancements to Multi-Blade milling let you do the following:

- Blade Finish using Flank Milling (swarfing).
- Use the enhanced tool axis controls to obtain a “machinable” tool path.
- Use the enhanced **Automatic** tool axis option to better orient the tool so that it avoids collisions.
- Increase the depth of cut in roughing passes by adding intermediate levels for embedded cuts.
- Finish cut the adjacent sides of opposing blades using the **Opposing Side** option.
- Specify the stock of adjacent blades in **Blade Finish** and **Blend Finish** operations.



Opposing Sides

Where do I find it?

Application	Manufacturing
Location in dialog box	(**Opposing Side**) **Blade Finish** or **Blend Finish** operation dialog box® **Drive Method** group® **Edit** ® **Blade Finish Drive Method/Blend Finish Drive Method**dialog box® **Cut Periphery** group® **Sides to Cut**

Multi-Blade axis control enhancements

What is it?

The **Automatic** tool axis option is enhanced to change the lead angle as necessary to avoid collisions.

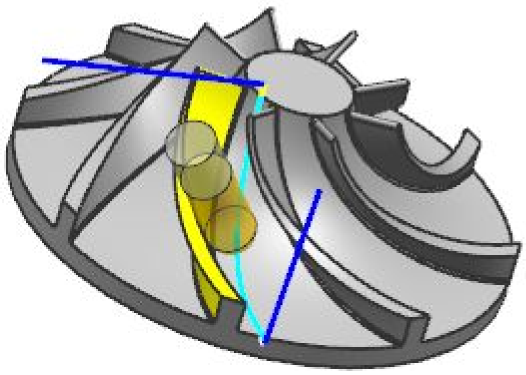
Enhancements to Multi-Blade axis controls let you do the following:

- Select the new **Swarf Blade** tool axis option to align the tool automatically to the blade profile.
  - o You can use the swarf orientation as a base for any **Multi Blade** operations.
  - o The **Swarf Blade** option is not sensitive to UV alignment, and accepts multiple UV patches.
  - o To finish cut blades using the side of the tool for swarf cutting or flank milling, set the **Number of Cuts** option to 1, or more if needed.
- Explicitly control the lead angle for zig zag motions:
  - o From the leading to the trailing edge.
  - o From the trailing edge to the leading edge.

- Specify a **Minimum Lead Angle** value to prevent heal digging.
- Use the enhanced controls for rotary axis limits in the following ways:
  - o Prevent tool paths that cannot be milled by your machine using the **Max Angle From Part Axis** option.
  - o Guide NX to by-pass singularities by using the **Min Angle From Part** option.

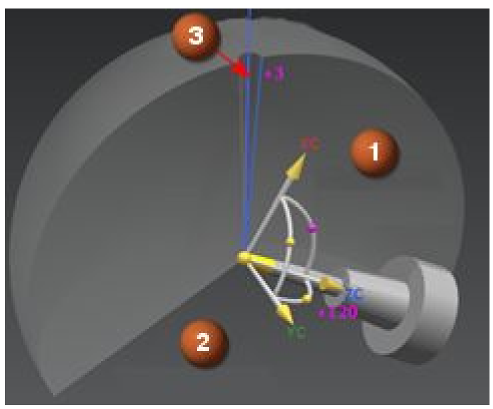
    When the tool is almost aligned with the rotary axis, a small angular change in tool orientation may require a nearly 180 degree motion of the table. This condition is called a *singularity*.

Swarf Blade



Axis = **Swarf Blade**, **Cut Levels**, **Number of Cuts** = 1

Rotary axis limit control



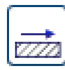
1: Motion is allowed within the **Min Angle From Part Axis** and **Max Angle From Part Axis** limits.

2: Motion is prohibited beyond the **Max Angle From Part Axis** limit.

3: Motion near the pole area is high-risk. The machine can do this motion, but the machining results are not optimal.

This limit is set by the **Min Angle From Part Axis** option.

Where do I find it?

Application	Manufacturing
Location in dialog box	[Multi Blade] operation dialog box: (**Swarf Blade**) **Tool Axis** group (Lead angles, minimum lead angle) **Tool Axis** group® **Automatic**® **Edit** ® **Automatic** dialog box® **Tool Axis** group (Rotary axis control) **Path Settings** group® **Cutting Parameters** ® **Cutting Parameters** dialog box® **Tool Axis Control** tab® **Machine Tool Limits** group

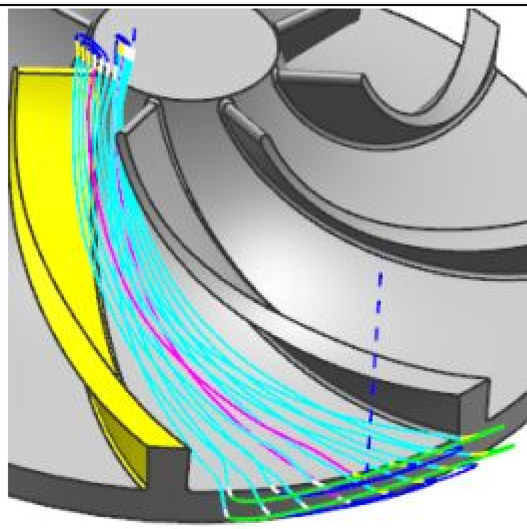
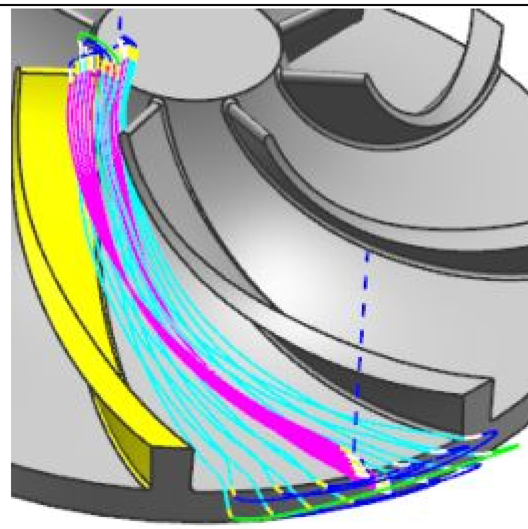
Multi-Blade milling cut level enhancements

What is it?

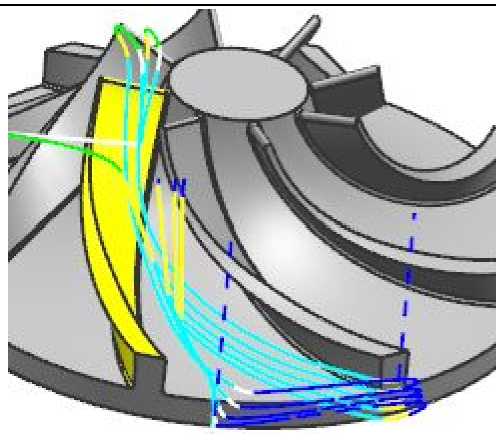
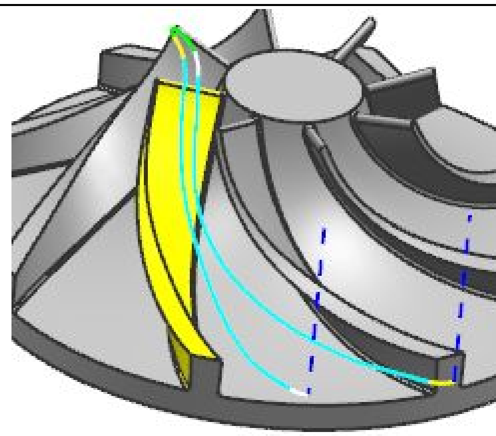
Enhancements to cut level control let you do the following:

- Add multiple depths to slotting passes of roughing operations using the new **Reduce Depth per Cut when Embedded** option. This option:
    - o Reduces the load on the first cut.
    - o Allows increase depth for the main cut levels.
    - o Dramatically reduces roughing time.
- Create incomplete cut levels and issue warning messages, or omit incomplete levels as needed using the new **Incomplete Levels** option.

Reduce Depth per Cut when Embedded

	
Reduce Depth per Cut when Embedded	**Reduce Depth per Cut when Embedded** **Number of Intermediate Cuts** = 5

Incomplete cut levels

	
Incomplete Levels = **Output and Warn**	**Incomplete Levels** = **Omit**

Where do I find it?

Application	Manufacturing
Location in dialog box	[Multi Blade] operation dialog box® **Path Settings** group® **Cut Levels** ® **Cut Levels** dialog box® **Depth Options** group

Multi-Blade milling check stock

What is it?

You can now control check stock for Multi-Blade milling operations in the following ways:

- Specify a **Check Stock** value.

- Select a stock option to apply to adjacent blades. You can apply the blade stock or the check stock.

  For Multi-Blade **Blade Finish** and **Blend Finish** operations, the geometry of an adjacent blade is considered check geometry unless you are using the **Opposing Sides** cut pattern.

Where do I find it?

Application	Manufacturing
Location in dialog box	[Multi Blade] operation dialog box® **Path Settings** group® **Cutting Parameters** ® **Cutting Parameters** dialog box® **Stock** tab® **Stock** group

contour profile enhancements

Contour Profile variable axis profiling enhancements

What is it?

Enhancements to the variable-axis **Contour Profile** operation give you better control of the operation and improve the robustness of the tool path. You also have the ability to use this operation as part of a complete finishing flow. You can now:

- Extend the tool path along cutouts in wall areas, using the new **Extend Distance** option.

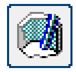
- Specify how to handle wall cutouts where the start and end of cut extensions leave a gap, using the new **Across Wall Gaps** option.

- Specify a start point when machining contoured parts with a closed cross section.

- Control axial shift of the tool, using the new **Ring Height** option.

- Overlap the wall finish passes and the floor finish passes to generate a cleanup operation that removes uncut blank material left by the previous tool.

Where do I find it?

Application	Manufacturing
Toolbar	**Insert® Create Operation** 
Menu	**Insert® Operation**
Location in dialog box	**Type** list® **mill_multi_blade® Operation Subtype** group® **Contour Profile** 

Contour Profile Ring Height

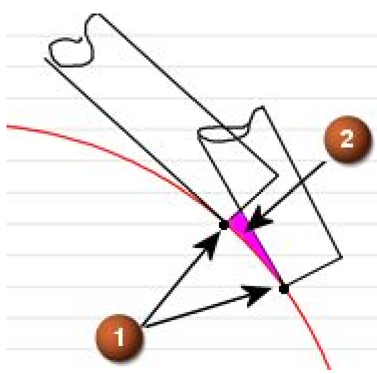
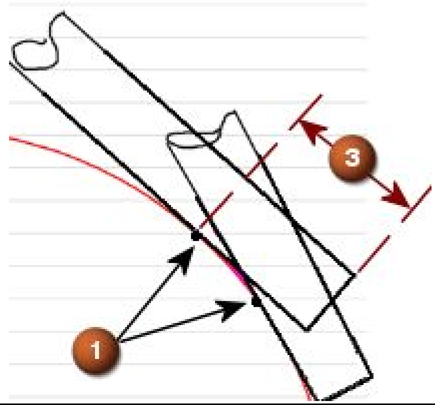
What is it?

Variable-axis contour profile operations cut with the side of the tool. The new **Ring Height** option lets you shift the tool along its axis to reduce scallops between cuts. The contact point on the part is not affected.

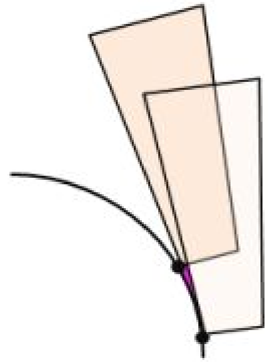
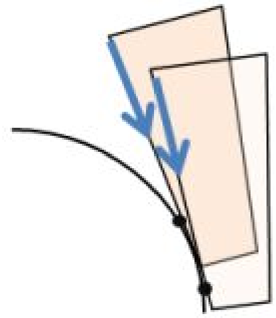
You can equalize tool wear with the **Variable** option. Specify a top and a bottom distance to control how far the tool can move.

NX applies the axial shift to all cutting motions as well as to extensions and across void motions. If the axial shift could gouge the part, NX lifts the tool along its axis or silhouette to prevent the gouge.

A positive value pushes the tool down. Negative values are not supported.

1 = contact points, 2 = scallop, 3 = **Ring Height**, **Distance** value	
	
Ring Height = **Constant** **Distance** = 0	**Ring Height** = **Constant** **Distance** = 12

Tapered tools are supported.

	
Ring Height = **Constant** **Distance** = 0	**Ring Height** = **Constant** **Distance** = 12

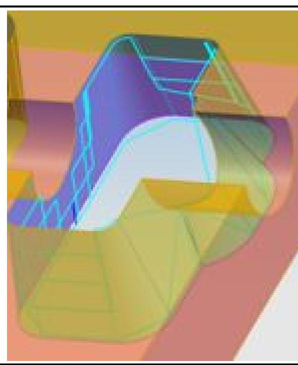
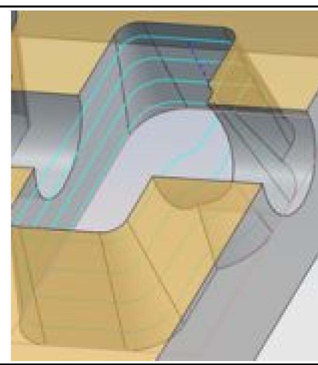
Where do I find it?

Application	Manufacturing
Prerequisite	**Contour Profile** operation
Location in dialog box	**Contour Profile** operation dialog box® **Drive Method** group® **Edit** ® **Contour Profile Drive Method** dialog box® **Contact Position** group

Contour Profile wall gap handling

What is it?

In previous releases, cuts that lost contact with the wall would create a poor quality tool path. Now, NX can generate cuts that follow the trimmed portions of the wall.

	
Previous releases	NX 8.5

New **Across Wall Gaps** options are available. Use these options to control the tool path behavior across wall gaps in the following ways:

- Specify **Extend Distance** values to extend the start or end of cuts.

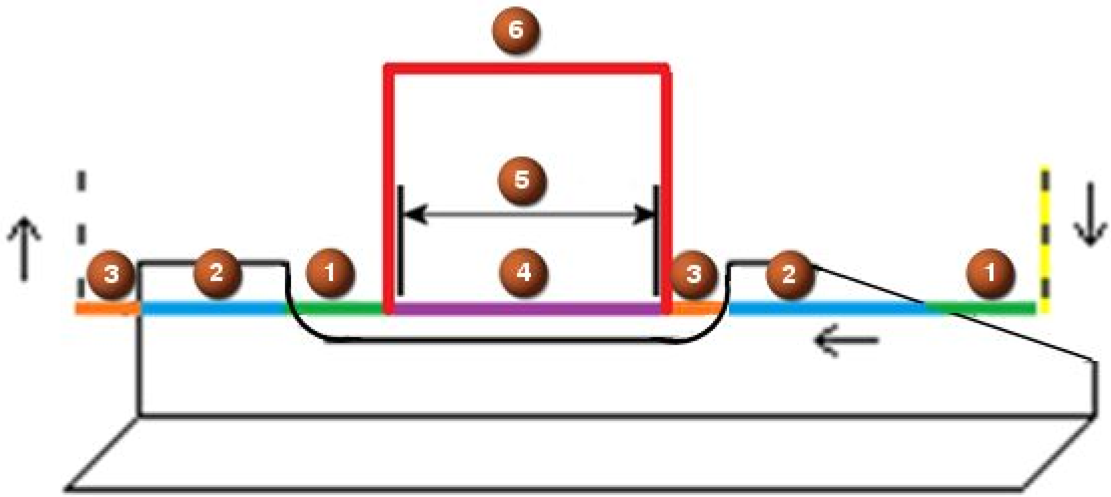
- Set the **Motion Type** option to **Cut** or **Stepover**.

  The **Stepover** motion lets you apply a faster feed rate.

- Specify when to apply a Non Cutting Moves (NCM) cycle.

  The Non Cutting Moves cycle can be more efficient for large wall gaps. Specify a **Minimum Distance** value to control when NX applies the Non Cutting Moves **Traverse Type** option selected for within regions.

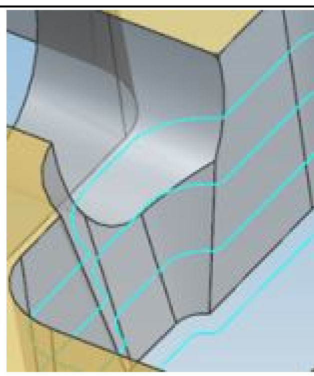
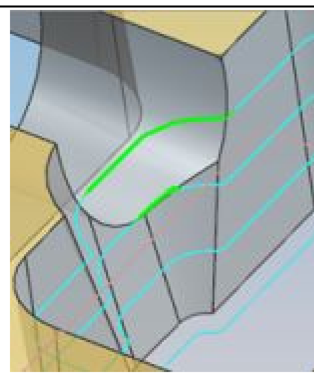
The following schematic diagram illustrates the **Across Wall Gaps** options.

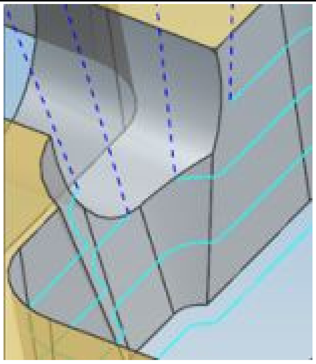
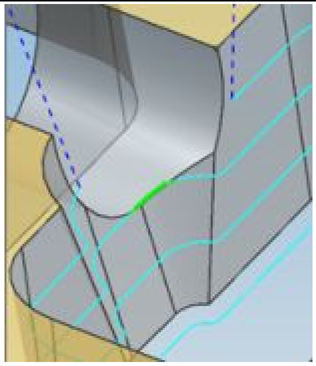
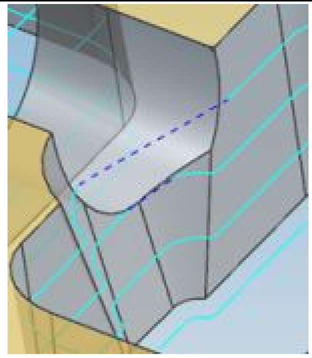


Wall gap with start and end of cut extensions

1: Start of cut extension	4: Wall gap: **Cut** or **Stepover** motion
2: Wall cut	5: **Minimum Distance** to be considered
3: End of cut extension	6: NCM cycle to traverse the gap

Examples

 **Motion Type** = **Cut**	 **Motion Type** = **Stepover**

 **Motion Type** = **Cut** **Minimum Distance** = 100	 **Motion Type** = **Stepover** **Minimum Distance** = 0.5	 **Motion Type** = **Cut** **Minimum Distance** = 0.0
Traverse Type = **Clearance**		**Traverse Type** = **Direct**

Where do I find it?

Application	Manufacturing
Prerequisite	**Contour Profile** operation
Location in dialog box	**Contour Profile** operation dialog box® **Drive Method** group® **Edit** ® **Contour Profile Drive Method** dialog box: (**Motion Type**, **Minimum Distance**) **Across Wall Gaps** group (**Extend Distance**) **Start of Cut** group, **End of Cut** group

Contour Profile start points

What is it?

You can now specify a start point when machining contoured parts with a closed cross section.

Automatic start point

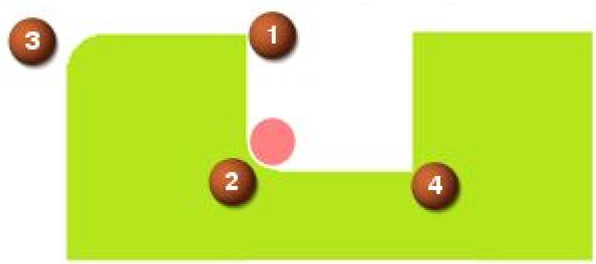
The **Automatic** option starts the tool path at one of the following locations, listed in order of priority:

1. A sharp convex corner.

2. The midpoint of a concave corner if the tool fits into the curvature.

3. The midpoint of a convex corner.

4. A sharp concave corner or fillet where the tool does not fit.



User Defined start point

If none of the **Automatic** option locations are acceptable, you can manually specify a start point location using the **User Defined** option. NX starts the tool path at the closest safe location to the manually specified start point.

Where do I find it?

Application	Manufacturing
Prerequisite	**Contour Profile** operation
Location in dialog box	**Contour Profile** operation dialog box® **Drive Method** group® **Edit** ® **Contour Profile Drive Method** dialog box® **Start of Cut** group

Contour Profile cleanup passes

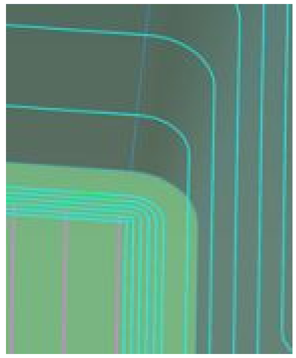
What is it?

Use the new **Only Cut Along Wall and Floor** option to generate a cleanup operation that removes uncut blank material left by the previous operation. This option overlaps the wall finish passes and the floor finish passes.

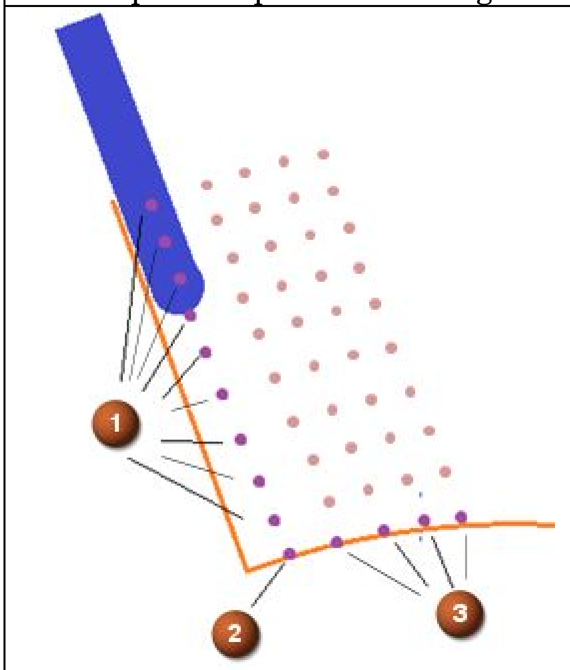
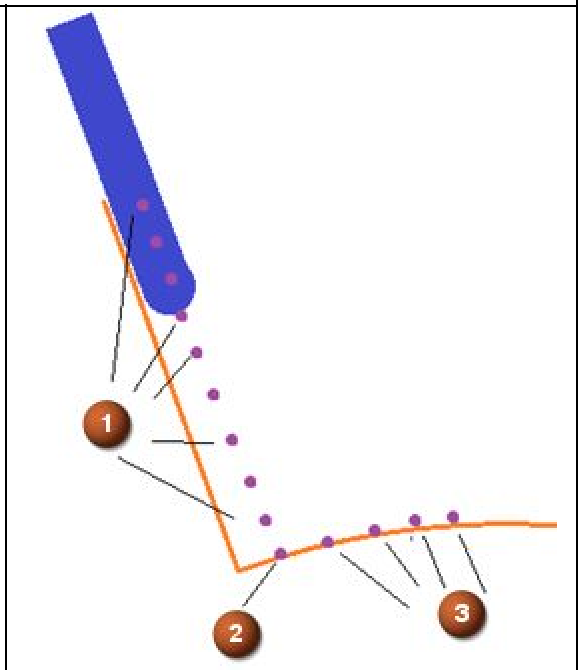
NX generates the tool passes in the following order:

1. Depth passes that cut from the top of the part down to the dual contact point of the current tool.

2. Side passes that cut away from the wall.

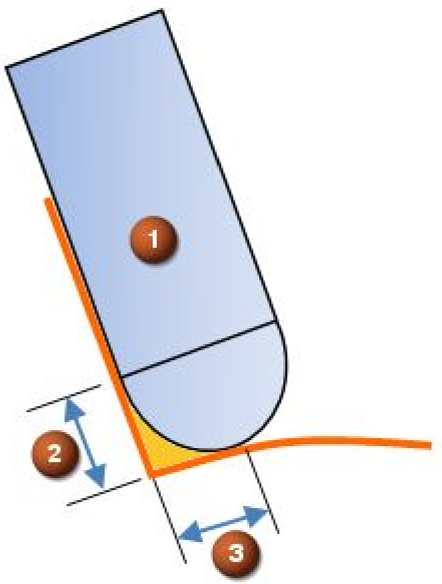
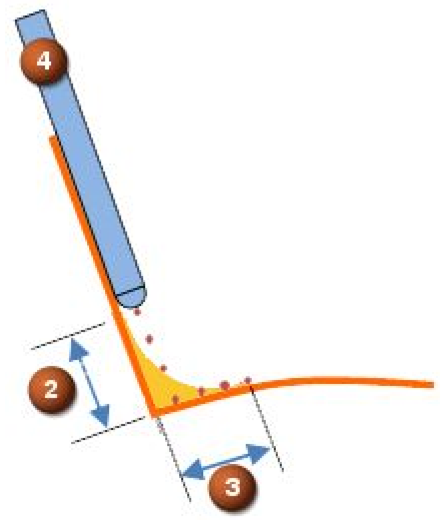


Only Cut Along Wall and Floor

<table>
<tr><td colspan="2">1: Multiple depth passes following the floor
2: Dual contact point of the current tool
3: Multiple side passes following the wall</td></tr>
<tr><td></td><td></td></tr>
<tr><td>Only Cut Along Wall and Floor</td><td>Only Cut Along Wall and Floor</td></tr>
</table>

Adjusting stock depth to support horizontal corner cleanup

With the appropriate side and depth stocks, you can use the **Contour Profile** operation to clean up horizontal corners.

<table>
<tr><td colspan="2">1: Tool from the previous operation
2: Depth Stock Offset
3: Side Stock Offset
4: Current tool</td></tr>
<tr><td></td><td></td></tr>
<tr><td>Depth stock and side stock determined from previous operation</td><td>Contour Profile clean up passes: 4 depth passes + 4 side passes</td></tr>
</table>

Where do I find it?

Application	Manufacturing
Prerequisite	**Contour Profile** operation
Location in dialog box	**Contour Profile** operation dialog box® **Path Settings** group® **Cutting Parameters** ® **Cutting Parameters** dialog box® **Multiple Passes** tab® **Output Passes** group

Turning enhancements

Siemens Sinumerik 840D thread cutting CYCLE97

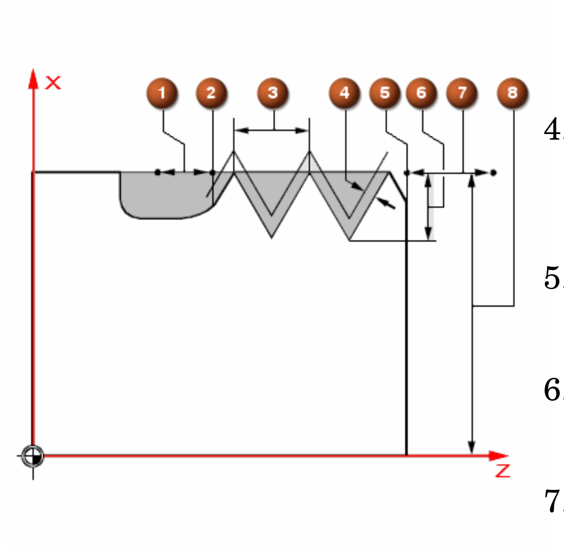
What is it?

The NX turning processor supports output for the Siemens Sinumerik 840D's `CYCLE97` thread cutting cycle in these operation types:

- Thread OD

- Thread ID

Mapping between Sinumerik variables and MOM variables



1. ROP
   Exit path
   0
2. FPL
   End point of thread in the longitudinal axis
   `mom_lathe_thread_crest_line_end(2)`
3. PIT
   Pitch: 0.001 to 2000 mm
   `mom_turn_thread_pitch_lead`
   `mom_number_of_starts`
4. FAL
   Finishing allowance
   `mom_total_depth_finish_passes_increment`
   `mom_total_depth_finish_passes_number_of_passes`
5. SPL
   Start point of thread in the longitudinal axis
   `mom_lathe_thread_crest_line_start(2)`
6. TDEP
   Thread depth
   `mom_turn_cycle_total_depth`
7. APP
   Run-in path
   0
8. DM1
   Diameter of thread at start point
   `mom_lathe_thread_crest_line_start(0)`
   DM2
   Diameter of thread at end point
   `mom_lathe_thread_crest_line_end(0)`

The machining cycle type maps the Sinumerik option **VARI** to the MOM variable `mom_turn_cycle97_machining_type`.

Programming Notes

ROP and APP are always zero in the template.

The MOM procedure `MOM_load_lathe_thread_cycle_params` supports cycle97 events in your postprocessor when the **Machine Cycle** option is specified.

This procedure:

- Returns 1 if the **Machine Cycle** option is specified and all variables are set successfully.

- Returns 0 otherwise.

Call the procedure `MOM_skip_handler_to_event` to skip event handling until an event or motion type that you specify is encountered during processing.

The following variables are available if you create similar user-defined threading cycles:

- `mom_lathe_thread_clearance_start`
- `mom_lathe_thread_clearance_end`
- `mom_lathe_thread_root_line_start`
- `mom_lathe_thread_root_line_end`

Why should I use it?

The post created by Post Builder using the Siemens 840D option has increased support for Siemens controllers.

On the shop floor, the machine operator can adjust a single line of code, for example to change the correct finishing allowance for the contour. To make the same change with conventional output, you must recalculate many lines of code using NX Post.

Where do I find it?

Application	Manufacturing
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 in dialog box	**Program & Tool Path® Custom Command® PB_CMD_map_cycle97_param**

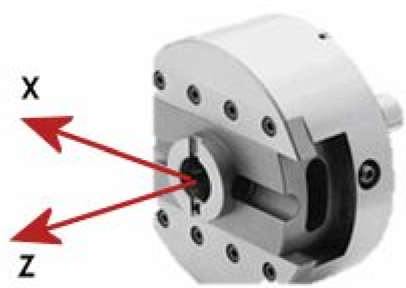
Variable work plane for turning tools

What is it?

A turning tool was formerly restricted to cut within the lathe work plane, or in some cases, in planes parallel to the lathe work plane.

To accommodate contouring heads, you can now create one or more operations that use an arbitrary work plane. Set the **MCS Spindle Group** list to **Operation** in the tool dialog box.

A typical contouring head supported by the new **Operation** option has an NC-controlled X-axis. The spinning head moves along the Z-axis to cut a stationary workpiece. You can position the workpiece or the head to an arbitrary position before cutting starts. The cutting plane is defined in the operation.



Where do I find it?

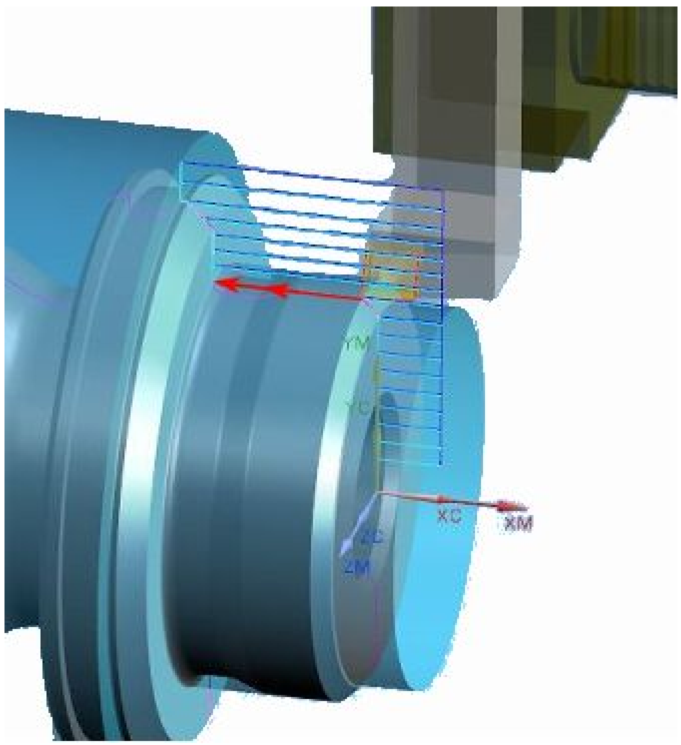
Application	Manufacturing
Prerequisite	You must be creating or editing a turning tool.

Toolbar	**Insert**® **Create Tool**
Menu	**Insert**® **Tool**
Location in dialog box	**More** tab® **Work Coordinate System** group® **MCS Spindle Group** list ® **Operation**

Roughing tool path enhancement

What is it?

When you create a **Rough OD** or **Rough ID** operation, NX produces a more efficient tool path which does not re-cut areas that have already been machined.



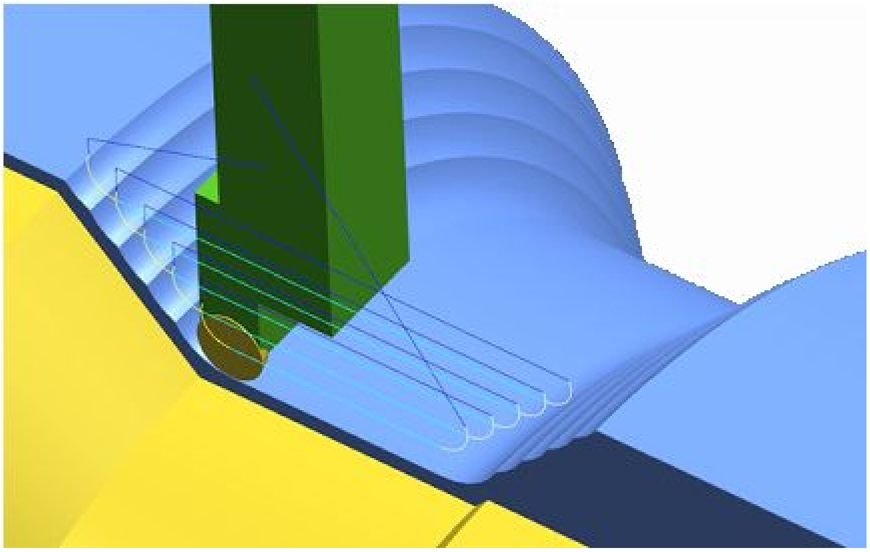
Where do I find it?

Application	Manufacturing
Prerequisite	• Turning module • **Rough Turn OD** or **Rough Turn ID** operation
Location in dialog box	**Cut Strategy** group® **Strategy** list® **Linear Zig** or **Linear Zig Zag**

Collision avoidance in two-point tangent engage and retract

What is it?

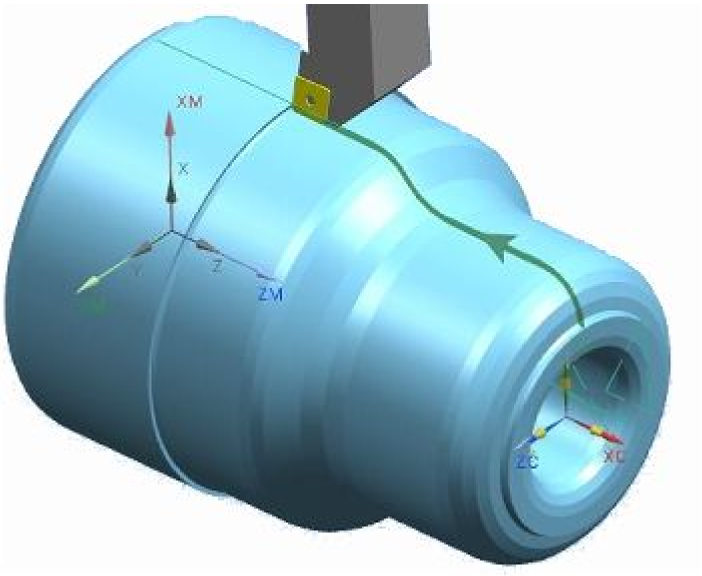
NX now automatically avoids collisions during stepovers in two-point engage and retract motions.



Blank Contour Zig

What is it?

Use the **Blank Contour Zig** cut strategy to better control roughing cut depths when you machine parts with irregular surfaces or parts made of difficult to machine materials. NX follows the contour of the blank and not the part.



Why should I use it?

This cut strategy gives you more efficient tool paths and extends your tool life.

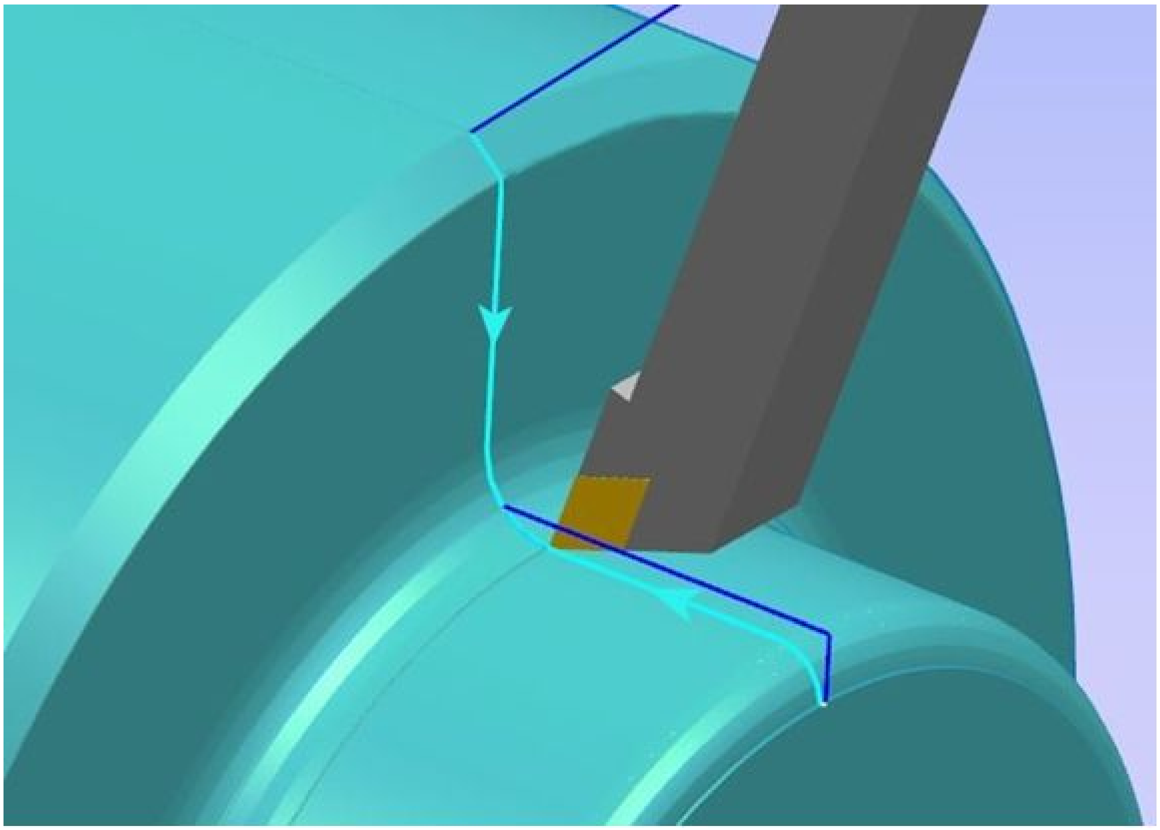
Where do I find it?

Application	Manufacturing
Toolbar	**Insert**® **Create Operation** 
Menu	**Insert**® **Operation**
Location in dialog box	Roughing operation dialog box® **Cut Strategy** group® **Strategy** list® **Blank Contour Zig**

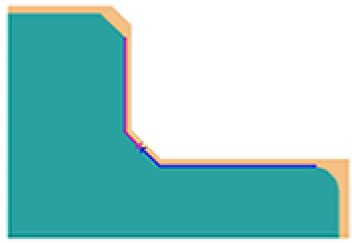
Finishing corners

What is it?

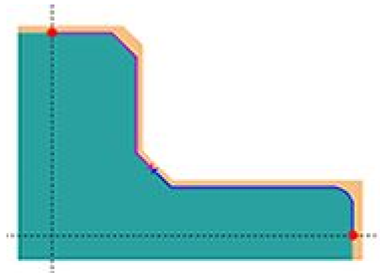
New **Extend at Start** options are available for machining into corners.



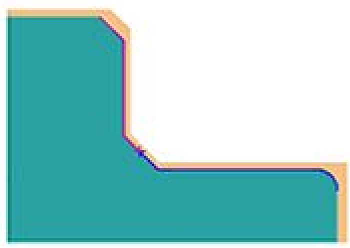
- **None** machines into a corner using two cut segments that cut towards the corner. One segment cuts along the diameter and the other segment cuts along the face.



- **To Containment Geometry** lets you specify a start and an end point on each cut segment, to contain the tool.



- **Include Adjacent Chamfers or Rounds** lets you add an adjacent chamfer or an adjacent blend to the start of a cut segment.



Where do I find it?

Application	Manufacturing
Prerequisite	A Turning finish operation that contains a chamfer, fillet or blend. **Cut Strategy** must be set to **Towards Corner**.
Menu	Finish operation dialog box® **Path Settings** group® **Extend at Start** list

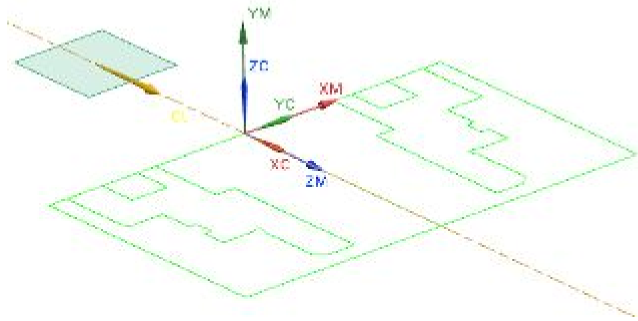
Lathe spindle and workplane display

What is it?

NX now displays the spindle and lathe workplane when you:

- Open the **MCS Spindle** dialog box

- Select the **MCS Spindle** parent group in the **Geometry** view of the **Operation Navigator**.



Why should I use it?

The spindle and work plane display makes it easier to understand the orientation of the new lathe workplane.

Where do I find it?

Application	Manufacturing
Prerequisite	Turning operation
Menu	**Operation Navigator**® **Geometry** view ® **MCS_SPINDLE**

Allow Selection of 2D IPWs

What is it?

The new **Allow Selection of 2D IPWs** option allows you to turn on or turn off the selectability of 2D IPW geometry.

Why should I use it?

If you turn this option on, you can select the 2D IPW geometry to measure distance, length, and so on. To ensure that the 2D IPW geometry is not accidentally selected when you select part geometry, turn the option off.

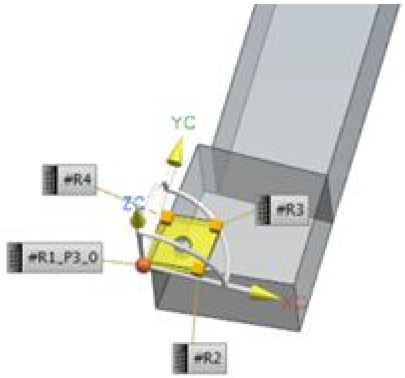
Where do I find it?

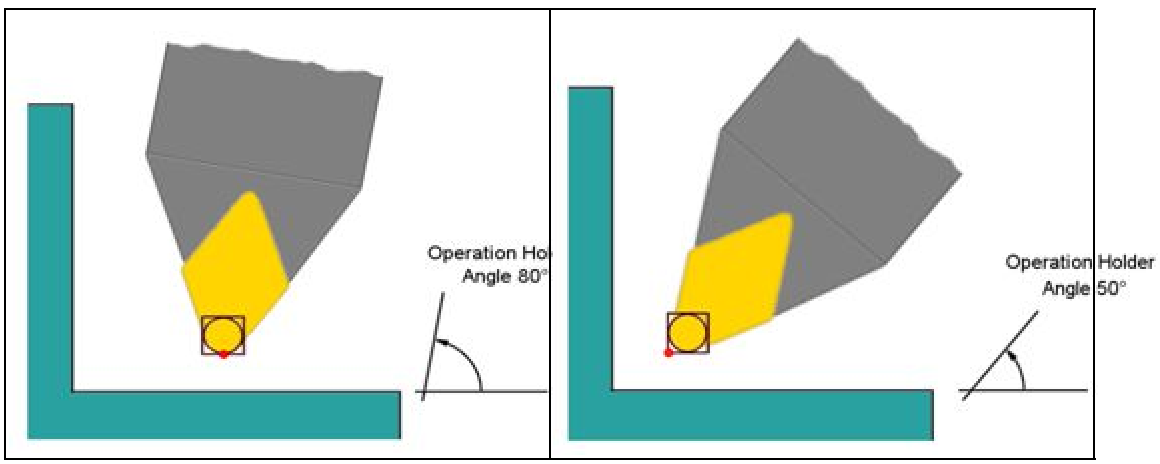
Application	Manufacturing
Toolbar	**Workpiece**® **Allow Selection of 2D IPWs**

Automatic option for tool tracking

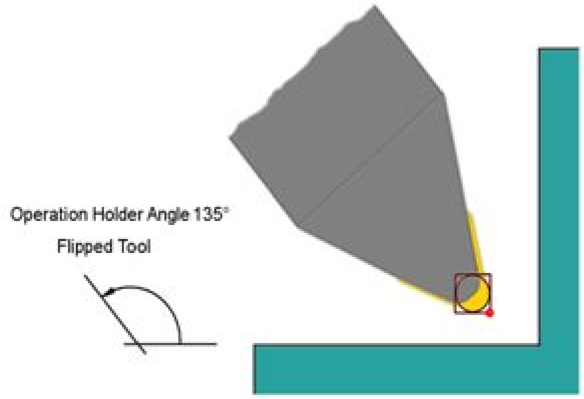
What is it?

The **Automatic** tool tracking point option has been added to the **Tracking Point** list. The **Automatic** point option automatically determines the location for the tracking point instead of the old way of manually selecting it.





The **Automatic** option also calculates the appropriate tracking point based on the tool holder angle and if the tool should get flipped around in the tool holder.



Note NX performs a mapping between the tracking point specified for the underlying tool in the tool definition dialog and the tracking point to be used with the resulting tool orientation of the operation in question.

Where do I find it?

Application	Manufacturing
Prerequisite	Rough or Finish turn operation **Reorient Tool Holder** check box must be selected
Menu	**Tool Orientation® Tracking Point**

Integrated simulation and verification — ISV

Tool head management

What is it?

You can mount tool heads to your machine tool model statically during a CAM session, or dynamically during simulation.

You must create heads in NX and submit them to the device library.

Why should I use it?

In simulation. you can check if your tool path is correct and detect collisions between heads, other machine components, and the part.

Column order in the Machine Tool Navigator

What is it?

You can now change the order of columns in the **Machine Tool Navigator**. You cannot move the first column, which is the **Name** column.

Why should I use it?

If you want to see a column in a narrow view of the navigator, you can move the column as close as the second position. For example, if you are making edits to holding systems, you can move the **Holding Systems** column from the last position to the second position.

Where do I find it?

Application	Manufacturing or Machine Tool Builder
Machine Tool Navigator	Right-click in the background® **Properties** Right-click in the background® **Columns**® **Configure**
Machine Tool Navigator Properties dialog box	**Columns** tab® select a column® or

Keep Assembly Constraints

What is it?

A new **Keep Assembly Constraints** option is available in the **Part Mounting** dialog box. When you replace a machine tool in a CAM setup, you can use this option to retain assembly constraints between the part, blank, or fixture and the machine tool model. Constraints between components of the machine tool assembly are not affected.

Why should I use it?

You need not recreate assembly constraints that position the part on the machine.

Where do I find it?

Application	Manufacturing
Operation Navigator	**Machine Tool** view® right-click the machine node® **Edit**
Location in dialog box	Machine parameters dialog box® **Retrieve Machine From Library** ® **Library Class Selection** dialog box® Select class of machine to retrieve® **Search Result** dialog box® select a machine® **Part Mounting** dialog box® **Placement** group® **Positioning** list® **Keep Assembly Constraints**

Remove Machine

What is it?

You can now remove a machine tool that was added to a CAM setup, using the new **Remove Machine** option in the **Machine Tool** view of the **Operation Navigator**.

Where do I find it?

Application	Manufacturing
Prerequisite	A machine tool must be present in the CAM setup part.
Operation Navigator	Right-click the machine tool node® **Remove Machine**

Simulation enhancements

What is it?

Simulation is enhanced to improve support of integrated simulation and verification, or ISV:

- When you start ISV, NX checks for the following critical errors before simulation starts:
  - Tools with the same number, for any simulation.
  - Tools with the number 0 for simulation of an external code file.
  - Spindle speed of 0, inherited tool number of 0, and inherited fixture offset of 0, for simulation of an internal operation.
- When you define a simulation collision pair that includes the in-process workpiece or IPW, the IPW is no longer automatically enabled . NX simulates that collision pair only when you turn on the IPW.

Why should I use it?

Advance error detection saves you simulation time.

You can control whether collision checking for the IPW is performed by turning the IPW on or off.

Where do I find it?

Collision checking

Application	Manufacturing
Toolbar	**Operations® Simulate Machine**
Operation Navigator	Right click an operation or program® **Tool Path® Simulate**
Simulation control Panel	**Simulation Settings** group® **Options® Simulation Options** dialog box® **Collision Detection** group® **Collision Detection** list® select **On® Specify Collision Pairs**

IPW control

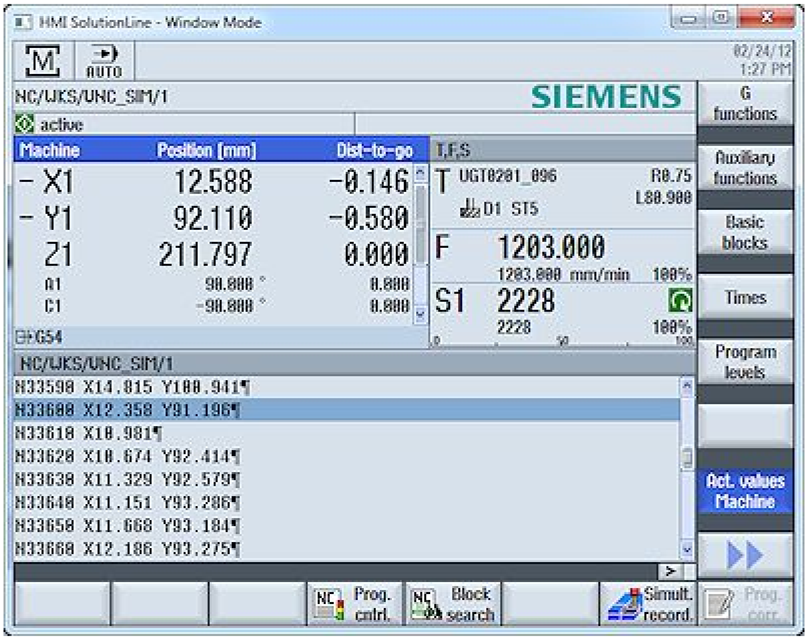
Simulation Control Panel	**Simulation Settings** group® **Options**
Location in dialog box	**Simulation Options** dialog box® **In Process Workpiece** group® **In Process Workpiece** list® **Off** or **Motion Based** or **Length Increment**.

VNCK machine tool simulation

You can simulate machine code from NX or external sources using the actual logic of a Sinumerik controller and an NX kinematic model of your machine tool. When you use the **Simulation Control Panel** dialog box, if VNCK is installed and configured for the current machine tool, you can change the **Visualization** option in the **Animation** group to **Machine Code Simulate**.

When you use **Machine Code Simulate**, the **Machine Control Panel** group, is added to the dialog box, and the **Human Machine Interface** dialog box is displayed. You can control the virtual machine and receive the same messages as you would on a real machine.

The example shows an HMI window while a program is running.



Where do I find it?

Application	Manufacturing
Prerequisite	You must have the HMI and VNCK software installed, you must obtain a set of machine and controller compatible CSE files, and you must obtain a suitable sram file that contains a memory image of a Sinumerik controller configured for your machine tool.
Toolbar	**Operations® Simulate Machine** 
Operation Navigator	**Tool Path® Simulate**
Location in dialog box	**Simulation Control Panel** dialog box® **Animation** group® **Visualization®** **Machine Code Simulate**

Sinumerik collision avoidance setup tool

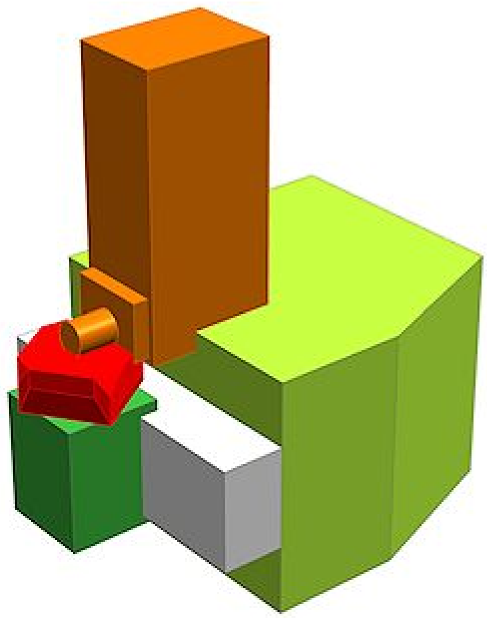
SinuCAST, or Sinumerik collision avoidance setup tool, is NX-integrated application used by machine tool manufacturers to create a simplified machine tool model which can be loaded into a Sinumerik controller. The model is created at a level of geometric and kinematic detail that allows the controller's collision avoidance detection system to work efficiently in real time.

You can perform these operations:

- Create an assembly tree consisting of imported part files, imported STL bodies, or primitives.

- Remove geometric features such as slots, holes, chamfers, and fillets from any component.
- Perform kinematics definition using the NX Machine Tool Builder.
- Define global stock, clearance value, or minimum distance for the collision avoidance geometry of the machine tool.
- Configure the size and shape of basic cutting tools.
- Detect potential collisions and collisions during simulated machine tool movement.

This shows a collision being detected between a simplified model of the machine and a facet representation of the workpiece.



SinuCAST toolbar

(1) Primitives list: Lets you create the NX features **Block**, **Cylinder**, and **Sphere**.
(2) Simplify Bodies: Lets you replace a selected body with **Nothing**, **Convex Hull**, **Bounding Sphere**, **Bounding Block**, **Bounding Cylinder**.
(3) Synchronous Modeling Toolbar: Shows or hides the NX **Synchronous Modeling** toolbar.
(4) Export Sinumerik SPF: Opens a dialog box where you control the export of geometry, protection areas, and kinematics.

Where do I find it?

Application	SinuCAST
Prerequisite	You must obtain the SinuCAST license.
Menu	**Start® All Applications® SinuCAST**

NX Post

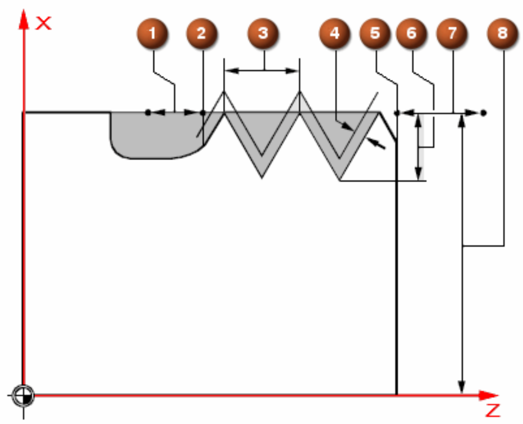
Siemens Sinumerik 840D thread cutting CYCLE97

What is it?

The NX turning processor supports output for the Siemens Sinumerik 840D's `CYCLE97` thread cutting cycle in these operation types:

- Thread OD
- Thread ID

Mapping between Sinumerik variables and MOM variables



1. ROP
   Exit path
   0
2. FPL
   End point of thread in the longitudinal axis
   `mom_lathe_thread_crest_line_end(2)`
3. PIT
   Pitch: 0.001 to 2000 mm
   `mom_turn_thread_pitch_lead`
   `mom_number_of_starts`
4. FAL
   Finishing allowance
   `mom_total_depth_finish_passes_increment`
   `mom_total_depth_finish_passes_number_of_passes`
5. SPL
   Start point of thread in the longitudinal axis
   `mom_lathe_thread_crest_line_start(2)`
6. TDEP
   Thread depth
   `mom_turn_cycle_total_depth`
7. APP
   Run-in path
   0

8. DM1
   Diameter of thread at start point
   `mom_lathe_thread_crest_line_start(0)`
   DM2
   Diameter of thread at end point
   `mom_lathe_thread_crest_line_end(0)`

The machining cycle type maps the Sinumerik option **VARI** to the MOM variable `mom_turn_cycle97_machining_type`.

Programming Notes

ROP and APP are always zero in the template.

The MOM procedure `MOM_load_lathe_thread_cycle_params` supports cycle97 events in your postprocessor when the **Machine Cycle** option is specified.

This procedure:

- Returns 1 if the **Machine Cycle** option is specified and all variables are set successfully.
- Returns 0 otherwise.

Call the procedure `MOM_skip_handler_to_event` to skip event handling until an event or motion type that you specify is encountered during processing.

The following variables are available if you create similar user-defined threading cycles:

- `mom_lathe_thread_clearance_start`
- `mom_lathe_thread_clearance_end`
- `mom_lathe_thread_root_line_start`
- `mom_lathe_thread_root_line_end`

Why should I use it?

The post created by Post Builder using the Siemens 840D option has increased support for Siemens controllers.

On the shop floor, the machine operator can adjust a single line of code, for example to change the correct finishing allowance for the contour. To make the same change with conventional output, you must recalculate many lines of code using NX Post.

Where do I find it?

Application	Manufacturing
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 in dialog box	**Program & Tool Path® Custom Command® PB_CMD_map_cycle97_param**

Post Builder

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The NX turning processor supports output for the Siemens Sinumerik 840D's `CYCLE97` thread cutting cycle in these operation types:

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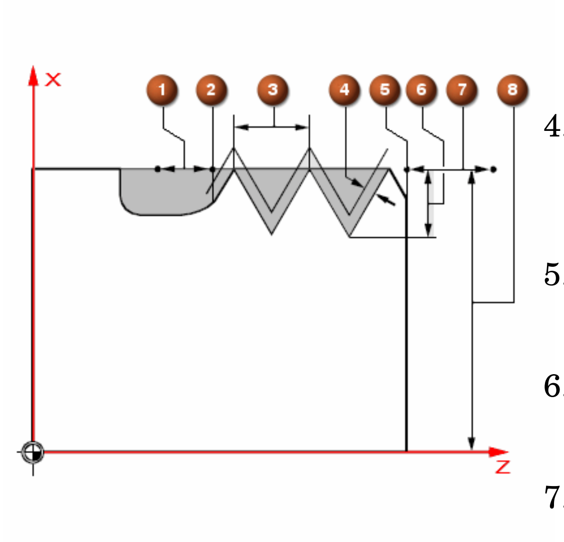
Mapping between Sinumerik variables and MOM variables



1. ROP
   Exit path
   0
2. FPL
   End point of thread in the longitudinal axis
   `mom_lathe_thread_crest_line_end(2)`
3. PIT
   Pitch: 0.001 to 2000 mm
   `mom_turn_thread_pitch_lead`
   `mom_number_of_starts`
4. FAL
   Finishing allowance
   `mom_total_depth_finish_passes_increment`
   `mom_total_depth_finish_passes_number_of_passes`
5. SPL
   Start point of thread in the longitudinal axis
   `mom_lathe_thread_crest_line_start(2)`
6. TDEP
   Thread depth
   `mom_turn_cycle_total_depth`
7. APP
   Run-in path
   0
8. DM1
   Diameter of thread at start point
   `mom_lathe_thread_crest_line_start(0)`
   DM2
   Diameter of thread at end point
   `mom_lathe_thread_crest_line_end(0)`

The machining cycle type maps the Sinumerik option **VARI** to the MOM variable `mom_turn_cycle97_machining_type`.

Programming Notes

ROP and APP are always zero in the template.

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This procedure:

- Returns 1 if the **Machine Cycle** option is specified and all variables are set successfully.
- Returns 0 otherwise.

Call the procedure `MOM_skip_handler_to_event` to skip event handling until an event or motion type that you specify is encountered during processing.

The following variables are available if you create similar user-defined threading cycles:

- `mom_lathe_thread_clearance_start`
- `mom_lathe_thread_clearance_end`
- `mom_lathe_thread_root_line_start`
- `mom_lathe_thread_root_line_end`

Why should I use it?

The post created by Post Builder using the Siemens 840D option has increased support for Siemens controllers.

On the shop floor, the machine operator can adjust a single line of code, for example to change the correct finishing allowance for the contour. To make the same change with conventional output, you must recalculate many lines of code using NX Post.

Where do I find it?

Application	Manufacturing
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 in dialog box	**Program & Tool Path® Custom Command® PB_CMD_map_cycle97_param**

Postprocessor access to junctions

What is it?

You can now access junctions, such as the machine zero junction, in a postprocessor by using Tcl procedures. An example is included in the file *simulation_ini.tcl* file in the Sinumerik postprocessor folder of the new *sim16_mill_headchange* machine in the `${UGII_CAM_LIBRARY_INSTALLED_MACHINES_DIR}` folder.

```
/******************************************************************************/
/*
* Syntax:  MOM_ask_machine_zero_junction_name
*
*    This command returns the name of the Machine Zero Junction.
*    Global variable mom_sim_result will also contain the result.
*/
/******************************************************************************/

/******************************************************************************/
/*
* Syntax:  MOM_ask_init_junction_xform jct_name
*
*    This command fetches the transformation matrix of the given Junction (name)
*    w.r.t the absolute coordinate system (ACS), where
*
*       mom_sim_result  : matrix (list of 9)
*       mom_sim_result1 : origin (list of 3)
*
*
*    Important note: Transformation returned is of the initial state of the
*                    Junction before the machine makes any movement at all.
*/
/*----------------------------------------------------------------------------*/
```

Why should I use it?

You can use these procedures to make your posts more versatile for machine tool simulation.

Where do I find it?

Application	**Post Builder**
Location in dialog box	**Program & Tool Path® Custom Command® Import**

Feature-based Machining

Large tool database support

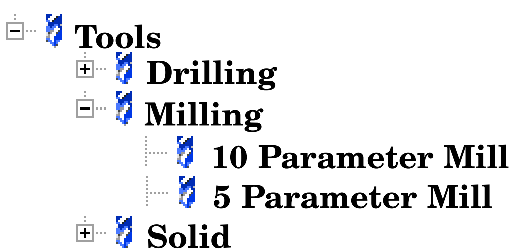
The Machining Knowledge Editor now supports the following:

- The same hierarchical tool class customization supported in NX.
- Attribute inheritance from a class to its sub-classes. Inherited attributes include the following:
  - o Cutter diameter
  - o Cutter length

  You can use the cutter diameter and length attributes in the **Set as Cutter Length** and **Set as Cutter Diameter** commands.
- Display of the NX tool name, `UI_NAME`, as well as the database name. You can use `UI_NAME` in the Machining Knowledge Editor rule conditions.
- Tool definitions from Teamcenter Classification.

  The tool definitions support the Machining Resource Library (MRL).

The example illustrates an MKE customization display that shows NX hierarchy and tool names.



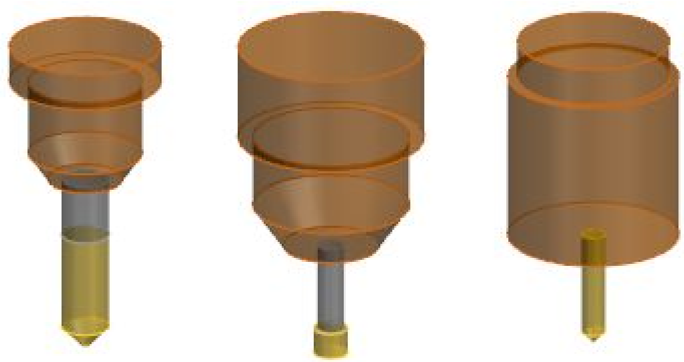
Where do I find it?

Application	Machining Knowledge Editor
Prerequisite	A *NX30435 — NX Feature Based Machining Author* license.

Automatic tool selection enhancement

What is it?

When you create operations for features, to reduce the number of tool changes, NX first tries to select library tools that are already used in the part file. If tools that match the requirements are not available in the part file, then NX retrieves them from the library.



Where do I find it?

Application	Manufacturing
Prerequisite	You must start Manufacturing with the **cam_general**, **feature_machining**, or **hole_making** cam session configuration.
Machining Feature Navigator	Right-click the features® **Create Feature Process**
Toolbar	**Feature** toolbar® **Create Feature Process** 

Teach Features and Find Features enhancements

What is it?

Datum CSYS support

Feature-based machining uses a datum CSYS to position and orient features. Feature teaching now fully supports all CSYS subtypes which reference a point, an edge or a face of the feature geometry.

Annotations support

Feature-based machining uses dimension annotations (PMI) to define feature attributes such as length and depth.

The **Teach Features** command now supports the following additional PMI dimension types which reference feature geometry:

- An inferred length dimension between a plane and a straight edge.
- An angle dimension between a plane and a straight edge.
- A dimension to the centerline of a cylindrical face.

The **Find features** command now recognizes the length and angle dimension tolerances that correspond to the annotation PMI.

Support for threaded hole or boss cylindrical faces

The **Teach Features** command now supports symbolic thread annotation.

The **Find features** command now recognizes symbolic threads and can extract the thread attributes. The recognition fails if a symbolic thread is not found where one is expected on a feature, or if there are multiple threads on a single feature.

Where do I find it?

Application	Manufacturing
Prerequisite	You must start Manufacturing with the **cam_general**, **feature_machining**, or **hole_making** cam session configuration.
Machining Feature Navigator	Right-click in the background® **Teach Features** or **Find Features**
Toolbar	**Feature** toolbar® **Teach Features**  or **Find Features** 

Chapter

4 *CAE*

NX 8.5 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 8.0.1	NX 8.5
NX Nastran	Import ASCII (.dat)	8	8	8.5
	Import Binary (.op2)	8	8	8.5
	Export ASCII (.dat)	8	8	8.5
	Post-processing of Results (.op2)	8	8.1	8.5
MSC Nastran	Import ASCII (.dat)	2011.1	2011.1	2012.1
	Import Binary (.op2)	2011.1	2011.1	2012.1
	Export ASCII (.dat)	2011.1	2011.1	2012.1
	Post-processing of Results (.op2)	2011.1	2011.1	2012.1
Abaqus	Import ASCII (.inp)	6.10	6.10	6.10
	Import Binary	N/A	N/A	N/A
	Export ASCII (.inp)	6.10	6.10	6.10
	Post-processing of Results (.fil)	6.11	6.11	6.11-1
	Post-processing of Results (.odb)	6.10-EF1	6.11	6.11
ANSYS	Import ASCII (PREP7, CDB)	13	13	14
	Import Binary (.rst, .rth)	13	13	14
	Export ASCII (.inp)	13	13	14
	Post-processing of Results	13	13	14
LS-DYNA	Import ASCII	971R5.0	971R5.0	971R5.0
	Import Binary	N/A	N/A	N/A
	Export ASCII (.k)	971R5.0	971R5.0	971R5.0
	Post-processing of Results	971R5.0	971R5.0	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 7.5.4	NX 7.5.5.
NX Nastran	Import ASCII (.dat)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Import Binary (.op2)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Export ASCII (.dat)	6.1	7.0	7.0	7.1	7.1	7.1	8

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2	NX 7.5.3	NX 7.5.4	NX 7.5.5.
	Post-processing of Results	6.1	7.0	7.1	7.1	7.1	7.1	8
MSC Nastran	Import ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Import Binary (.op2)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Export ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Post-processing of Results	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
Abaqus	Import ASCII (.inp)	6.8-1	6.9–1	6.9–1	6.9-1	6.10	6.10	6.10
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9	6.9	6.9	6.10	6.10	6.10
	Post-processing of Results (.fil)	6.8-EF2	6.9.2	6.9.2	6.10-1	6.10-1	6.10-1	6.11-1
	Post-processing of Results (.odb)	6.8-EF2	6.9-EF1	6.9-EF2	6.9-EF2	6.10-EF1	6.10-EF1	6.10-EF1
ANSYS	Import ASCII (PREP7, CDB)	12	12.1	12.1	12.1	13	13	13
	Import Binary (.rst, .rth)	12	12.1	12.1	12.1	13	13	13
	Export ASCII (.inp)	12	12.1	12.1	12.1	13	13	13
	Post-processing of Results	12	12.1	12.1	12.1	12.1	12.1	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R3.2	971R3.2	971R3.2	971R3.2	971R3.2	971R3.2	971R3.2.1
	Post-processing of Results	N/A	N/A	971R3.2	971R3.2	971R3.2	971R3.2	971R3.2.1

NX 6 releases

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3	NX 6.0.4	NX 6.0.5
NX Nastran	Import ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Import Binary (.op2)	6.0	6.1	6.1	6.1	6.1	7.0
	Export ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Post-processing of Results	6.0	6.0	6.1	6.1	7.0	7.0
MSC Nastran	Import ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Import Binary (.op2)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Post-processing of Results	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
Abaqus	Import ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Post-processing of Results (.fil)	6.7-5	6.8-1	6.8-3	6.8-EF2	6.8-EF2	6.8-EF2
	Post-processing of Results (.odb)	N/A	N/A	N/A	6.8-EF	6.8-EF2	6.9-EF2
ANSYS	Import ASCII (PREP7, CDB)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Import Binary (.rst, .rth)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Export ASCII (.inp)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Post-processing of Results	11 SP1	11 SP1	11 SP1	11 SP1	12.0	12.1

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
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

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
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

Edit CAE model data in context

What is it?

Note This topic is currently under construction.

Beginning with this release, you can utilize many of the design-in-context workflows of NX assemblies when working with CAE models in Advanced Simulation. In CAE applications, these workflows are often referred to as *edit in context.*

In previous releases, for example, if you wanted to generate a mesh or edit an existing mesh, you would first need to change the displayed part to the FEM file containing the geometry. You can now set the work part independently of the displayed part.

In Advanced Simulation, edit in context supports many new workflows, including the following:

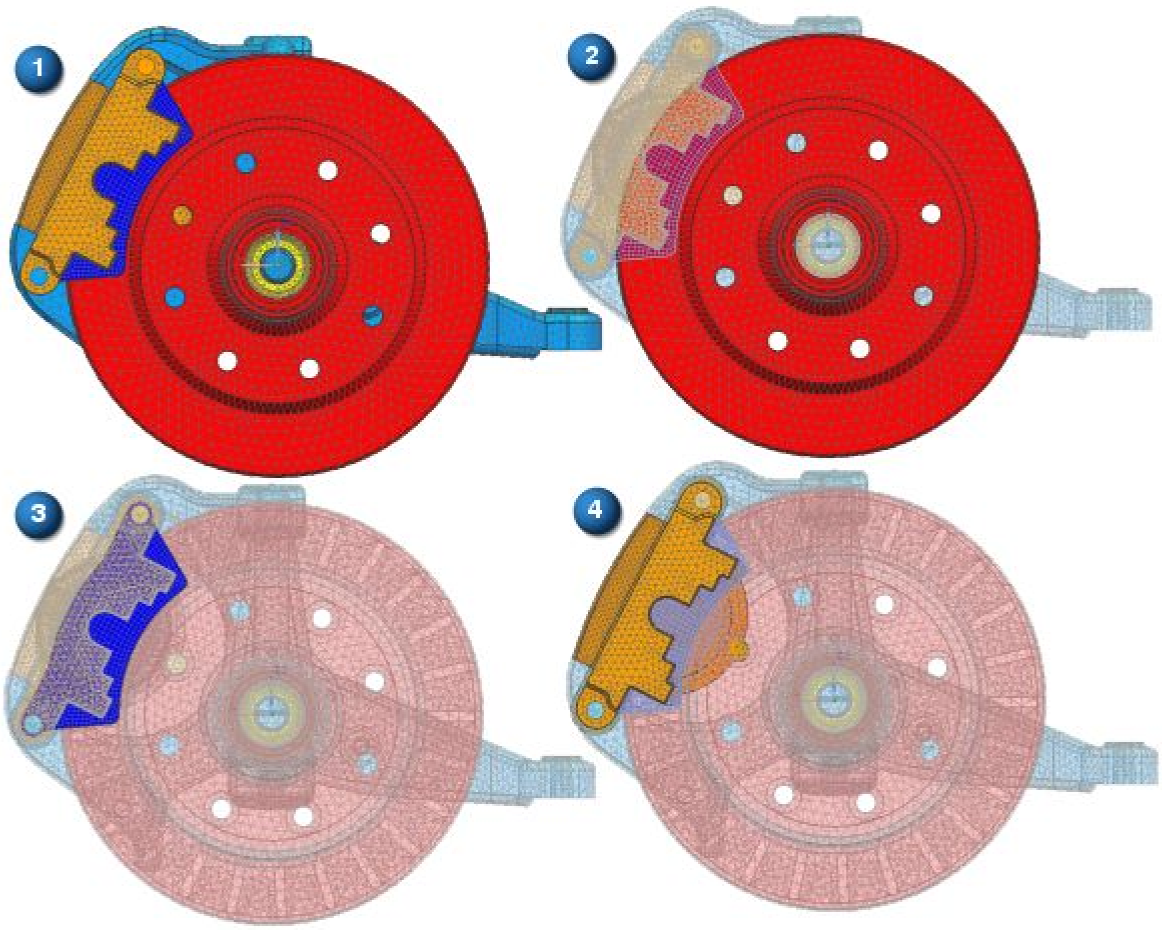
- With an assembly FEM displayed, make a component FEM the work part to create or edit meshes, clean up or modify polygon geometry, or modifiy physical and material properties.
- With an assembly FEM displayed, make a subassembly FEM the work part to resolve label conflicts or edit connection meshes.
- With a Simulation displayed, make the component FEM the work part to modify meshes or polygon geometry in the context of applied loads and boundary conditions.
- With a FEM displayed, make the idealized part the work part to create or modify geometry in the context of the mesh.

As with CAD assemblies, when the work part is different from the displayed part, the software indicates this in several ways:

- The work part and displayed part are indicated in the NX window title bar.

- Non-work-part nodes in the **Simulation Navigator** are dimmed.
- In the Graphics window, non-work parts are displayed as translucent.

The following image shows an assembly FEM with various component FEMs designated as the current work part.



An assembly FEM of a disk brake assembly. (1) The assembly FEM is both the displayed and the work part. (2) The disk is the work part. (3) The brake pad is the work part. (4) The caliper is the work part.

When you change the work part in the context of the displayed part, the menus, toolbars, and context menus change to reflect the commands available for that part.

Why should I use it?

In general, when working with large or complex CAE models, editing in context saves time and reduces resource consumption as compared to changing the displayed part each time.

When meshing components of an assembly FEM, editing in context enables you to quickly compare the relative mesh topologies where components connect. You can then edit your meshes using mesh controls, manual editing, and so on, to ensure high-quality connections between component FEMs and improve your model.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click a CAE model component node® **Make Work Part**

Model Check usability enhancements

What is it?

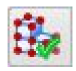
Note This topic is currently under construction.

This release includes substantial enhancements to model validation capabilities that were available from the **Model Checks** dialog box in previous releases. Beginning in this release, each of the **Model Checks** options is now a separate command.

- The user interface for each of these new commands has been improved to enhance usability.
- All commands now fully support journaling.
- All commands now include support **Smart Selector** selection options. For example, you can now select elements by their feature angle or
- The **Element Quality Checks** command now has extended capabilities, including support for solver specific quality checks. For more information, see Element quality check enhancements.

The following table shows the new commands:

Command	Icon	Description
Element Quality Checks		Lets you evaluate the qu elements in your model.
Element Edges		Lets you display any 2D elements with free edges you display any non-mani edges.
2D Element Normals		Lets you evaluate the co of the 2D element norma model.

Command	Icon	Description
Duplicate Nodes		Lets you check your mod duplicate nodes and merg those nodes together.
Duplicate Elements		Lets you check your mod duplicate elements and d those duplicates.
Element Material Orientation		Lets you display the mat orientation of 2D and 3D your model.
Adjust Node Proximity to CAD		Lets you adjust the proxi nodes created on polygon the underlying CAD geon
CAE Model Consistency		Lets you evaluate the po geometry in your model consistency issues.
Detect Interference/Clearance		Lets you detect regions of within your model.
Finite Element Model Summary		Lets you print an inform summary of the entities i file.
Simulation Summary		Lets you print an inform summary of the entities Simulation file.
Model Setup Check		Lets you evaluate wheth model is ready to solve.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file active
Toolbar	**Advanced Simulation**
Menu	**Analysis® Finite Element Model Check**

Element quality check enhancements

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes substantial enhancements to the element quality checks capabilities.

- Solver specific and element type specific quality checks
- Ability to specify both warning and error limits for a given quality check.
- Output group support
- New check for evaluating the length of elements

Solver and element type specific quality checks

You can use the new **Element Quality** command to evaluate the quality of elements in your model based upon the specific quality criteria used by your solver. In previous releases, the **Element Shapes** option in the **Model Check** dialog box only evaluated the quality of elements based on a general set of criteria.

Solver-specific quality checks are supported for the following solver environments:

- NX Nastran
- MSC Nastran
- Abaqus
- ANSYS

  Note The ANSYS solver-specific checks in NX are consistent with the quality checks in the standard ANSYS solver. They are not consistent with the quality checks used by the ANSYS Workbench platform.

- LS-DYNA.

For example, if you are working in the ANSYS solver environment, you can specify **Parallel Deviation** value limits for elements both with and without midside nodes.

Additionally, you can select different element quality criteria and specify different quality threshold values depending on the element's type. For example, in the NX Nastran environment, you can specify different **Aspect Ratio** values for different types of elements, such as tetrahedrons and pyramids.

Ability to set both warning and error limits

For a given quality check, such as **Aspect Ratio**, you can now specify separate **Warning Limit** and **Error Limit** values.

- Use the **Warning Limit** options to specify quality threshold values at which your solver issues a warning message.

- Use the **Error Limit** options to specify the threshold value at which your solver issues an error.

You can also use the **Warning Color** and **Error Color** option to specify different colors to use to display elements that violate these limits.

Output group support

You can use the new **Output Settings** options in the **Element Quality Checks** dialog box to have the software automatically place certain elements into a new group, named OUTPUT GROUP. This group will overwrite an existing OUTPUT GROUP.

You can choose to create a group of:

- Elements that fail any of the specified **Warning Limit** values.
- Elements that fail any of the specified **Error Limit** values.
- Elements that fail both the specified **Warning Limit** and **Error Limit** values.

New check for evaluating the length of elements

The **Element Quality Checks** dialog box now has a **Calculate Element Edge Size** option that allows you to calculate the minimum and maximum element edge size among the selected elements.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file active
Toolbar	**Advanced Simulation® Element Quality** 
Menu	**Analysis® Finite Element Model Check® Element Quality**

Essentials role for Advanced Simulation

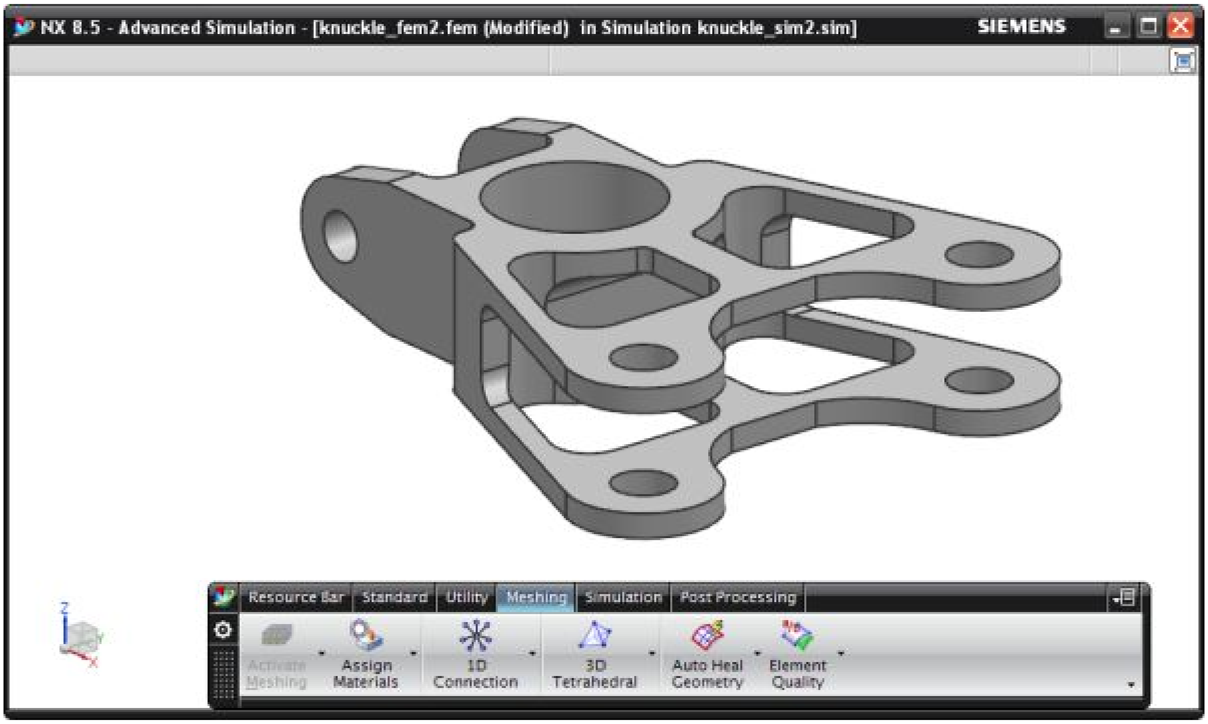
What is it?

Note This topic is currently under construction.

The Essentials role for Advanced Simulation has been enhanced to provide a focused, task-based environment for analyzing component parts.

With the Essentials role active, when you create a new FEM and Simulation file, the FEM is the displayed part, and the commands related to meshing the part are available. After you mesh the part and assign a material and physical properties, you can click the **Activate Simulation** button to make the Simulation file the work part, where you can add constraints and loads and solve the model.

This role is designed to be used with NX in full-screen display mode, which displays the toolbars as tabs, and more clearly guides you through the workflow of defining the analysis.



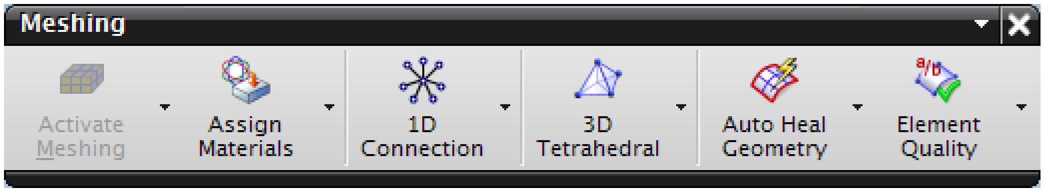
Essentials role in Advanced Simulation, shown in full-screen display mode

This role works only when you have a Simulation with a single associated FEM.

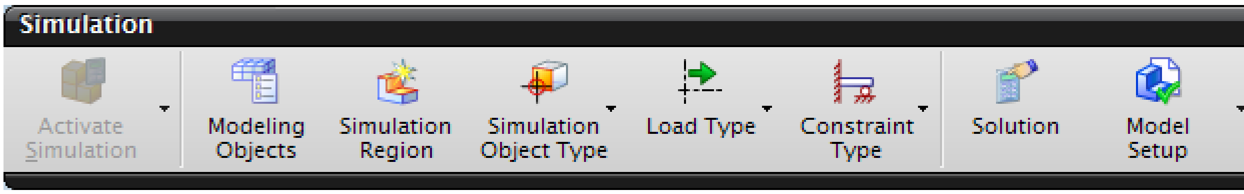
New Meshing and Simulation toolbars

With the Essentials role, only the core commands for meshing and defining boundary conditions are available, arranged on two new toolbars titled **Meshing** and **Simulation**.

- The **Meshing** toolbar contains the commands that are available when the FEM is the work part. It also contains commands that are available when either the Simulation file or FEM is the work part. The **Activate Meshing** button on this toolbar sets the FEM to the work part and makes the commands available.



- The **Simulation** toolbar contains the commands that are available when the Simulation file is the work part. It also contains commands that are available when either the Simulation file or FEM is the work part. The **Activate Simulation** button on this toolbar sets the Simulation file to the work part and makes the commands available.



The **Post Processing** toolbar is also available. Most of the ancillary NX toolbars are hidden, so you can better concentrate on the process of model preparation, meshing, applying boundary conditions, and post processing.

At any time, you can display the **Advanced Simulation** toolbar to use the full set of Advanced Simulation commands. This toolbar is hidden by default.

Where do I find it?

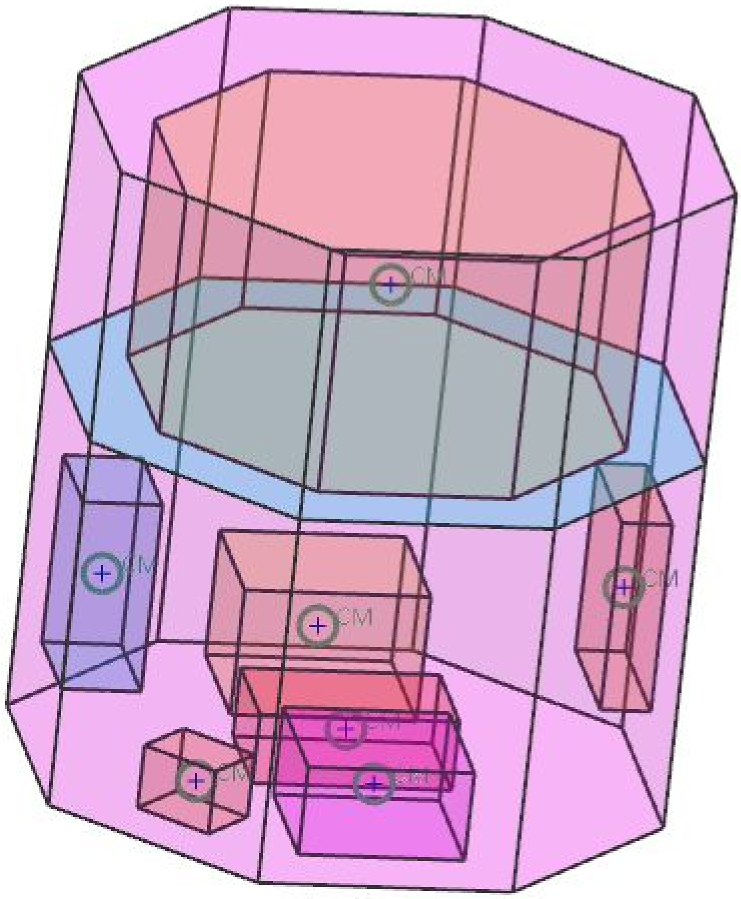
Application	Advanced Simulation
Roles navigator	**Essentials** 
Menu	**View® Full Screen**

Concentrated mass graphical representation

What is it?

Note This topic is currently under construction.

You can now choose from over 20 graphical symbols to represent 0D concentrated mass elements in your model.



NX Nastran CONM2 elements with Thick Circle symbol type

Where do I find it?

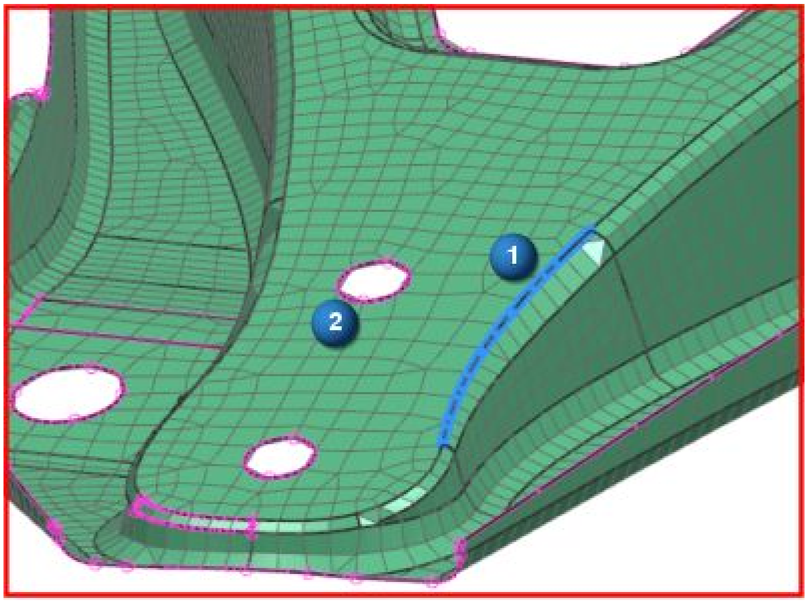
Application	Advanced Simulation
Simulation Navigator	Right-click a 0D mesh or mesh collector® **Edit Display**

Display stitched edges

What is it?

Note This topic is currently under construction.

In addition to displaying the free edges of polygon geometry, you can now display the stitched edges in a different color. The new **Display Stitched Edges** option is available on the **Model Display** dialog box.



(1) stitched edge; (2) free edge

Where do I find it?

Application	Advanced Simulation

Additional CAE entities available in Show and Hide command

What is it?

Note This topic is currently under construction.

You can now show and hide additional types of CAE objects using the **Show and Hide** command. The following object types are now available:

Category	Available CAE object types
Polygon Bodies	Sheet body Solid body Midsurface body Face from mesh body Flow body
0D Meshes	Concentrated mass elements Distributed mass elements Heat body elements Node to ground 1D elements

Category	Available CAE object types
1D Meshes	Bar elements Beam elements Rod elements Rigid link elements Interpolation elements Spring elements 1D Contact elements 2D Contact elements Weld elements Edge-Face connection elements PLOTEL elements 1D Mass elements
2D Meshes	Triangular elements with 3 nodes Triangular elements with 6 nodes Quadrilateral elements with 4 nodes Quadrilateral elements with 8 nodes
3D Meshes	Tetrahedral elements with 4 nodes Tetrahedral elements with 10 nodes Tetrahedral elements with no mid-nodes Hexahedral elements with 8 nodes Hexahedral elements with 20 nodes Wedge elements with 6 nodes Wedge elements with 15 nodes Pyramid elements with 5 nodes Pyramid elements with 13 nodes Pyramid elements with no mid-nodes

Where do I find it?

Application	Advanced Simulation, Design Simulation

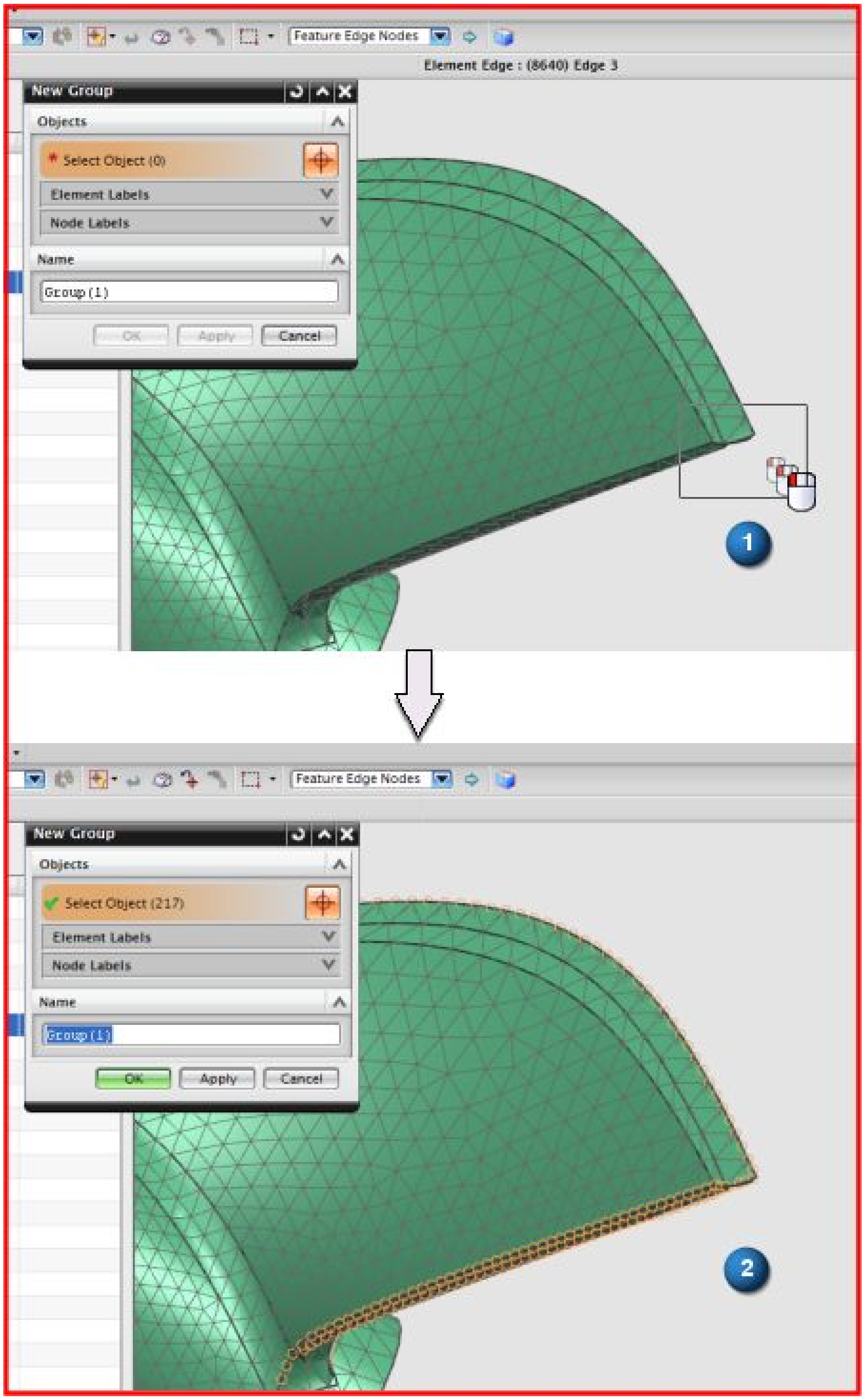
Toolbar	**Utility® Show and Hide**
Menu	**Edit® Show and Hide® Show and Hide**

Area-select during smart selection

What is it?

Note This topic is currently under construction.

You can now area-select seed elements or faces when using the selection **Method** options that are available in the Selection bar.



(1) Feature Edge Nodes method being used to area-select multiple edges; (2) resulting nodes are shown selected

The following methods support area-selection.

Selecting nodes and elements:

- Feature Angle Elements

- Related Elements
- Feature Angle Nodes
- Feature Edge Nodes
- By Group

Selecting polygon geometry:

- Tangent faces
- Adjacent faces
- Fillet faces
- Sliver faces
- Related faces
- Circular edges
- Tangent continuous edges

Where do I find it?

Application	Advanced Simulation, Design Simulation

Support for non-default assembly arrangements when creating CAE files

What is it?

Note This topic is currently under construction.

Beginning with this release, when you create a new FEM and Simulation file, or when you create a new assembly FEM, the assembly arrangement that is currently applied in the assembly will be used. In previous releases, the default assembly arrangement was used.

Where do I find it?

Application	Advanced Simulation

Fields enhancements

What is it?

Note This topic is currently under construction.

This release includes enhancements to fields functionality. These enhancements include:

- Surface spatial maps
- Table field interpolator enhancements
- **Table Field** and **Formula Field** dialog box improvements

Surface spatial maps

For table fields, you can now define surface spatial maps. Surface spatial maps let you map data that is modeled in a 3D spatial domain to a 2D surface.

Compared to the existing parameter plane method of mapping 3D data to a 2D surface, the new **Surface** subtype is easier to use and yields a mesh that can better conform to a complex surface.

To define surface spatial maps, you set the independent domain to **Cartesian**, **Cylindrical**, or **Spherical**. You set the **Subtype** to **Surface** or **Cloud**.

When the subtype is **Surface**, the 3D data maps directly to the 2D surface that you select. If the subtype is **Cloud**, the 3D data maps as a “cloud of points” and may not conform to geometry when you apply the field as a boundary condition. The **Surface** subtype can be a more effective way to map 3D data to a complex surface.

Table field interpolator enhancements

The current release includes the following enhancements to the **Table Fields** dialog box:

- If selected, the **Persistent Interpolator** checkbox lets you save interpolators across NX sessions. This can improve performance, since you no longer need to create an interpolator every time a part is opened, but only when the table data is changed.
- The new **Approximate Nearest Neighbor** interpolator enables improved performance in nearest neighbor searching.

  Given an error bound $\epsilon \geq \mathbf{0}$, the search algorithm returns **k** distinct points, such that for $\mathbf{1} \leq \mathbf{i} \leq \mathbf{k}$, the ratio between the distance to the **ith** reported point and the true **ith** nearest neighbor is at most $\mathbf{1} + \epsilon$.
- In previous releases, the **Inverse Weighted Distance** interpolator worked on the entire set of data points, which may be slow for very large tables. In this release, this interpolator now includes these new options to improve performance and accuracy: **All Points**, **Points within a Radius**, and **Nearest Points**. The interpolator uses the **Approximate Nearest Neighbor** library for improved performance while searching.

Table Field and Formula Field dialog box improvements

The **Table Field** and **Formula Field** dialog boxes have been modified to improve usability. The changes include the following:

- In the **Domain** group, the **Independent** and **Dependent** variables are no longer defined on tabs.

- The **Variables** table, **Units** list, **Bounds** options have been removed from the **Domain** group. The dialog box now contains a single table for independent variables. You can edit units and bounds values within the table.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation or FEM file.
Simulation Navigator	Right-click the **Fields** node® **New Field** [field type]

Material and physical properties

Material pedigree view

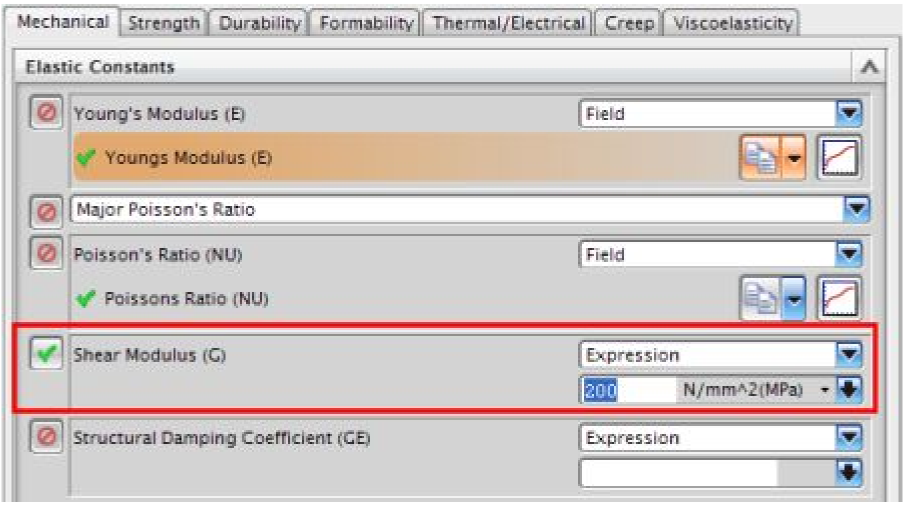
What is it?

Note This topic is currently under construction.

In the materials dialog boxes, such as **Manage Materials**, when you inspect or edit a material definition that has been copied from a library material, you can now determine which material properties are inherited from the parent material and which material properties have been changed.

- Properties that are inherited from the parent material are indicated with a new **Inherited Value** button.

- Properties that have been changed from the parent material record are indicated with a new **Overridden Value** button.



Shear Modulus has been overridden; all other properties are inherited

To revert a changed value to the original value, click the **Overridden Value**  button.

Where do I find it?

Application	All

Usability enhancements for materials

What is it?

Note This topic is currently under construction.

This release includes these new features that enhance the process of defining materials in NX.

- You can now create a material from the **Assign Material** dialog box. In previous releases, the **Create** command was available only on the **Manage Materials** dialog box.

- The material record now includes an **Alternate Name** property and **Sub-Category**.

Where do I find it?

Application	All

Material list filtering

What is it?

Note This topic is currently under construction.

You can create custom filters to control the list of materials that displays in the **Materials** dialog box. You can define rules to filter by characteristics of any material property, such as value ranges <, ≤, =, ≥, >, and Not Equal To.

For example, you can display all materials that meet this criteria:

- **Mass Density** < X to meet weight requirements
- **Young's Modulus** > Y to meet strength requirements
- **Category**, **Sub-Category**, and **Name** filters to consider only materials due to cost, availability, and environmental issues

Where do I find it?

Application	All
Menu	**Tools® Materials® Assign Materials** **Tools® Materials® Manage Materials** **Tools® Materials® Manage Library Materials**
Location in dialog box	Right-click in the **Material List® Create Filter**

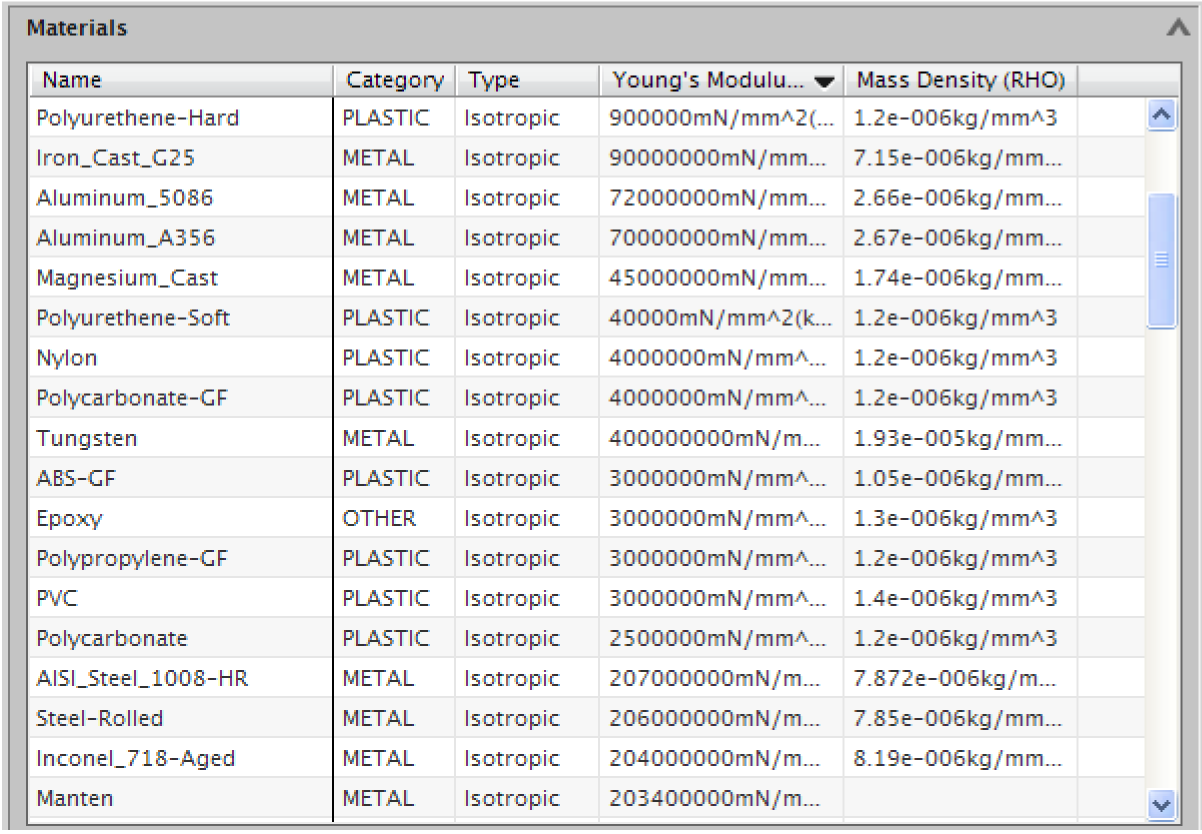
Material list columns

What is it?

Note This topic is currently under construction.

You can now add columns to the **Material List** view for any material property. You can also remove any column from the display.

For example, you could use this capability to sort the list of materials by **Young's Modulus** or **Mass Density**.


Materials

Name	Category	Type	Young's Modulu...	Mass Density (RHO)
Polyurethene-Hard	PLASTIC	Isotropic	900000mN/mm^2(...	1.2e-006kg/mm^3
Iron_Cast_G25	METAL	Isotropic	90000000mN/mm...	7.15e-006kg/mm...
Aluminum_5086	METAL	Isotropic	72000000mN/mm...	2.66e-006kg/mm...
Aluminum_A356	METAL	Isotropic	70000000mN/mm...	2.67e-006kg/mm...
Magnesium_Cast	METAL	Isotropic	45000000mN/mm...	1.74e-006kg/mm...
Polyurethene-Soft	PLASTIC	Isotropic	40000mN/mm^2(k...	1.2e-006kg/mm^3
Nylon	PLASTIC	Isotropic	4000000mN/mm^...	1.2e-006kg/mm^3
Polycarbonate-GF	PLASTIC	Isotropic	4000000mN/mm^...	1.2e-006kg/mm^3
Tungsten	METAL	Isotropic	400000000mN/m...	1.93e-005kg/mm...
ABS-GF	PLASTIC	Isotropic	3000000mN/mm^...	1.05e-006kg/mm...
Epoxy	OTHER	Isotropic	3000000mN/mm^...	1.3e-006kg/mm^3
Polypropylene-GF	PLASTIC	Isotropic	3000000mN/mm^...	1.2e-006kg/mm^3
PVC	PLASTIC	Isotropic	3000000mN/mm^...	1.4e-006kg/mm^3
Polycarbonate	PLASTIC	Isotropic	2500000mN/mm^...	1.2e-006kg/mm^3
AISI_Steel_1008-HR	METAL	Isotropic	207000000mN/m...	7.872e-006kg/m...
Steel-Rolled	METAL	Isotropic	206000000mN/m...	7.85e-006kg/mm...
Inconel_718-Aged	METAL	Isotropic	204000000mN/m...	8.19e-006kg/mm...
Manten	METAL	Isotropic	203400000mN/m...	

Where do I find it?

Application	All
Menu	**Tools® Materials® Assign Materials** **Tools® Materials® Manage Materials** **Tools® Materials® Manage Library Materials**
Location in dialog box	Right-click in the **Material List® Columns**

Displacement-based gasket material

What is it?

Note This topic is currently under construction.

This release includes a new displacement-based gasket material, named **Gasket Behavior Material**. This material is intended for ANSYS (the Nastran gasket material is strain based).

Where do I find it?

Application	All

New and changed material properties

What is it?

Note This topic is currently under construction.

This release includes several new material properties and changes to existing properties.

- Fatigue Strength and Ductility vs. Life fields
- Orthotropic Fatigue Strength and Ductility (previously available only for isotropic materials)
- Orthotropic Yield Strength and Ultimate Tensile Strength (currently available only for isotropic materials)
- Percent Reduction in Area (ASME parameter, Durability tab on isotropic and orthotropic materials)
- For Formability materials, you can now define Thickness, Bend Radius, Neutral Factors for Material Grades (each material grade is a separate material in NX)
- For Formability materials, you can now define Ship Insulation Material Properties
- Additional Viscoelastic support (the following properties are now unitless: OmegaG, OmegaK, Prony Series Constants, Compliance/Relaxation Data, and Volume Ratios)
- Units have been corrected on some gasket behavior material properties and on MATG EPL
- You can now define the Poisson's Ratio on all hyperelastic materials (Abaqus)
- The Poisson's Ratio has been separated into Major and Minor Poisson's Ratio (ANSYS)
- Other minor additions to hyperelastic materials (for example, Strain Energy Coefficient has been added to the Ogden material, Lamda M to Van Der Waals material, and other changes)
- For Creep, a Delta H term has been added that matches Abaqus
- For Fluid materials (MAT10), add the ability to define Density, Bulk Modulus, and Damping Coefficient as frequency dependent.

Where do I find it?

Application	All

Polygon geometry and geometry abstraction

New command for deleting polygon faces

What is it?

Use the new **Delete Face** command to delete selected polygon faces.

Note This topic is currently under construction.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file active
Toolbar	**Advanced Simulation® Delete Face**
Menu	**Insert® Model Cleanup® Delete Face**

Polygon geometry update improvements

What is it?

REVIEW NOTE

Issue: This topic is currently under construction.

This release includes improvements to the logic that the software uses to make updates to the polygon geometry in your FEM file. These improvements are designed to minimize the amount of polygon geometry that is affected to changes to the underlying CAD geometry. In previous releases, the polygon geometry update process occurred at the polygon body level. In this release, the polygon geometry process occurs at the polygon face level. As a result, the updates that occur are much more limited.

Now, during the polygon geometry update process:

- Fewer polygon faces are deleted and recreated.
- Manual modifications that you make to the polygon geometry are now preserved.
- Fewer mesh mating conditions and **Stitch Edge** recipes are recreated and replayed.
- Fewer meshes are deleted and recreated.

Meshing

New report detailing mesh update information

What is it?

Note This topic is currently under construction.

When you use the **FE Model Update** command to update the meshes in your FEM file, the software now generates a log file that provides detailed information about the mesh update process for your particular model. The log file lists, for example:

- The meshes that were updated.
- The order in which the meshes were updated.
- Any errors that occurred during the mesh update process.

Customer Default to control the generation of mesh update reports

This release also includes a new customer default that you can use to control the display of the mesh update information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® FE Model Update**
Menu	**Edit® Element® Split 1D Element**

New Split 1D Element command

What is it?

Note This topic is currently under construction.

You can use the new **Split 1D Element** command to subdivide selected 1D elements, such as beam, rod, or bar elements.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active assembly FEM file
Toolbar	**Element Operations® Split 1D Element**

Menu	**Edit® Element® Split 1D Element**

New mesh controls for welds and structuring the mesh around holes

What is it?

Note This topic is currently under construction.

You can now create the following new types of **Mesh Control** on 2D meshes:

- **Weld Row**
- **Mapped Holes**

Weld rows

The **Weld Row** type produces a structured (mapped) row of elements along a set of connected edges that define the location of a weld. With the **Weld Row** type, you can create the structured elements at a specified offset into the face or faces that are attached to the selected edges. This allows you to simulate the footprint of a weld in your 2D mesh along with the equivalent plate thickness in that region without requiring you to subdivide the underlying faces.

Structured meshes around holes

The **Mapped Holes** type produces structured rows of elements around specified holes or loops. With the **Weld Row** type, you can control:

- The number of layers (rows) of structured elements that the software creates.
- The spacing of those elements around the edge of the hole.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® Mesh Control**
Menu	**Insert® Mesh® Mesh Control**

Support for CWELD/CFAST connection parameters

What is it?

Note This topic is currently under construction.

This release now includes a new **CFAST/CWELD Connection Parameters** modeling object that provides full support for all the parameters on the `SWLDPRM` bulk data entry. You can use the options in the **CFAST/CWELD Connection Parameters** dialog box, for example, adjust tolerances and defaults when CFAST or CWELD creation fails. For example, you can control:

- Tolerance values that the software uses to determine the placement of the CWELD or CFAST element.
- Where the software calculates the displacements for CWELD or CFAST elements.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran as the specified solver
Toolbar	**Advanced Simulation® Bolt Pre-Load**
Menu	**Insert® Mesh® Mesh Control**

Ability to create two elements through the thickness in tetrahedral meshes

What is it?

Note This topic is currently under construction.

When you create a tetrahedral mesh on a part that contains regions with greatly varying thickness, you can now ensure that the software produces a minimum of two elements through the thickness of very thin regions. In the **3D Tetrahedral Mesh** dialog box, you can use the new **Minimum Two Element Through Thickness** option to ensure that the software creates a solid mesh at least two elements thick through very thin regions of a part.

The **Minimum Two Element Through Thickness** option is useful when you want to specify a large element size for the overall mesh but be assured that any thin-walled regions of an otherwise chunky part will have two elements through the thickness.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 3D Tetrahedral Mesh**
Menu	**Insert® Mesh® 3D Tetrahedral Mesh**

Transition algorithm improved for 2D and 3D free meshes

What is it?

Note This topic is currently under construction.

The software now uses an improved transition algorithm when you select the **Transition Element Size** option in the **2D Mesh** or **3D Mesh** dialog box. The **Transition Element Size** option controls the rate of element size change between coarse and fine regions of your mesh. The improved transition algorithm produces fewer elements, without sacrificing element quality, to accomplish the element size changes.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 2D Mesh**  or **3D Tetrahedral Mesh** 
Menu	**Insert® Mesh® 2D Mesh** or **3D Tetrahedral Mesh**

Additional edge to edge connection methods for Nastran

What is it?

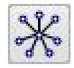
Note This topic is currently under construction.

In the **1D Connection** dialog box, when you select **Edge to Edge** from the **Type** list, you can select new strategies to connect the two edges in a Nastran model.

- With the **RBE2 and RBE3 to Element Edge** option, the software projects the nodes from the source edge onto the target edge. The software then creates an RBE2 between the two sets of nodes . The software also creates an RBE3 element between each projected node (this node becomes the dependent node) and the nodes on the target edge (these nodes become the leg nodes).

- Select **RBE2 and RBE3 to Element Face** option, the software projects the nodes from the source edge onto the target edge. The software then creates an RBE2 between the two sets of nodes. The software also creates an RBE3 element between each projected node (this node becomes the dependent node) and the nodes on the face of each element on the target edge (these nodes become the leg nodes).

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file active with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® 1D Connection** 
Menu	**Insert® Mesh® 1D Connection**

Manual element commands now ensure clockwise connectivity for solid elements

What is it?

Note This topic is currently under construction.

The commands on the **Element Operations** toolbar have been improved so that they now only create 3D elements that have node connectivity in a clockwise direction. Many solvers will not process a model that has 3D elements with counterclockwise node connectivity. This enhancement helps ensure that the software produces elements that fulfill this solver connectivity requirement.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations**

Modifying the type of elements

What is it?

Note This topic is currently under construction.

You can use the new **Element Modify Type** command to modify the type of selected elements without causing the software to regenerate the entire mesh. With the **Element Modify Type** command, you can now easily change the type of element used in a mesh to another type within the same element topology.

For example, you can use **Element Modify Type** to change an existing mesh of 1D bar elements to a mesh of 1D beam elements. You can also use **Element Modify Type** to change an existing mesh of 2D plate elements to a mesh of 2D plane strain elements. However, you cannot use **Element Modify Type** to change a mesh of 1D bar elements to a mesh of 2D plane elements.

You can use the **Element Modify Type** command to modify all types of meshes except for meshes that you created with the **2D Dependent Mesh** command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® Element Modify Type** 
Menu	**Edit® Element® Modify Type**

New command for projecting nodes

What is it?

Note This topic is currently under construction.

You can use the new **Node Project** command to manually project a node onto a selected edge or face. In the **Node Project** dialog box, you can project a node:

- Along a selected vector

- By the shortest distance to the selected edge or face.

With the **Node Project** command, you can choose to project the original node to a new location. You can also choose to copy the original node and project a new node onto the new location.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Node Operations® Node Project** 
Menu	**Edit® Node® Project**

New reflection plane creation methods supported for reflecting nodes and elements

What is it?

Note This topic is currently under construction.

When you use the **Node Reflect** or **Element Reflect** commands, there are new methods available for defining plane about which to reflect the nodes or elements.

- Use the new **By Three Points** option to select three points or nodes to define the reflection plane.

- Use the new **Triangular Element Face** option to use the face of a selected triangular element to define the reflection plane.

These new methods give you greater versatility in terms of how you specify the reflection plane.

Where do I find it?

Node Reflect

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Node Operations® Node Reflect**
Menu	**Insert® Node® Reflect**

Element Reflect

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® Element Reflect**
Menu	**Insert® Element® Reflect**

Properties are now preserved when you extrude or revolve elements

What is it?

Note This topic is currently under construction.

When you use the **Element Extrude** or the **Element Revolve** command to manually extrude or revolve elements from different meshes, the software now copies the properties from the source elements, such as the assigned material and display properties, to the newly created elements. In previous releases, the properties of the original elements were not preserved in the new elements created by the **Element Extrude** and **Element Revolve** commands.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® Element Extrude** or **Element Revolve**

Menu	**Insert**® **Element**® **Extrude** or **Revolve**

Ability to manually create elements directly on points

What is it?

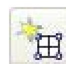
Note This topic is currently under construction.

You can now use the **Element Create** command to manually create elements directly on existing points. In previous releases, you could only create elements by selecting existing nodes.

Now when you manually create an element, if you select points to define the element, the software automatically:

- Creates nodes at those point locations.
- Associates those new nodes to the underlying points.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations**® **Element Create** 
Menu	**Edit**® **Element**® **Create**

New methods for translating nodes

What is it?

Note This topic is currently under construction.

This release includes several new methods for manually translating nodes. In the **Node Translate** dialog box, the **Method** list contains several new options:

- **By Aligning Vectors**
- **By Point to Point**
- **By Scale**

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active assembly FEM file
Toolbar	**Node Operations**® **Node Translate** 

Menu	**Edit® Node® Node Translate**

Boundary conditions

Glue and contact support enhancements for NX Nastran

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes significant enhancements to the support of NX Nastran glue and contact conditions in NX. These enhancements include:

- Glue support enhancements.
- Support for axisymmetric edge-to-edge contact.

Glue support enhancements

This release includes extensions and improvements to the glue connection capabilities.

- You can use the new **Edge-to-Edge Gluing** command to define glue connections between selected polygon or element edges. With this command, you can create glue connections between the edges of the following types of elements in NX:
  - o Shell elements CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR
  - o Axisymmetric shell elements CTRAX3, CQUADX4, CTRAX6, CQUADX8

  The elements whose edges are being glued must be in the same plane. For example, you cannot use the **Edge-to-Edge Gluing** command to glue the edges of two elements whose edges are perpendicular.

- When you use the **Edge-to-Surface Gluing** command to define glue connections between selected edges and selected surfaces, you can now select an edge region that is comprised of discontinuous edges. For more information, see Improvements to edge-based simulation regions (NX Nastran).

A glue connection is a simple and effective method to join meshes which are dissimilar. It correctly transfers displacement and loads resulting in an accurate strain and stress condition at the interface. A glue connection creates stiff springs or a weld like connection to prevent relative motion in all directions. The grid points on glued edges and surfaces do not need to be coincident.

Support for axisymmetric edge-to-edge contact

Use the new **Edge-to-Edge Contact** command to define contact conditions between selected polygon or element edges. With this command, you can define contact conditions between the following types of axisymmetric elements:

CTRAX3, CQUADX4, CTRAX6, CQUADX8 (the elements must be oriented in either the XZ or XY plane)

You can use the **Edge-to-Edge Contact** command in to define contact at selected edges in solutions 101, 103, 105, 111, and 112.

In NX Nastran, you use the `BCRESULTS` case control command to request edge-to-edge contact force, traction, and separation distance output. In Advanced Simulation, create a **Structural Output Requests** modeling object and select the **Enable BCRESULTS Request** option on the **Contact Result** tab.

Where do I find it?

Edge-to-Edge Gluing

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran as the specified solver environment
Toolbar	**Advanced Simulation® Edge-to-Edge Gluing** 
Simulation Navigator	Right-click **Simulation Objects® New Simulation Object® Edge-to-Edge Gluing**

Edge-to-Surface Gluing

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran as the specified solver environment
Toolbar	**Advanced Simulation® Edge-to-Surface Gluing** 
Simulation Navigator	Right-click **Simulation Objects® New Simulation Object® Edge-to-Surface Gluing**

Edge-to-Edge Contact

Application	Advanced Simulation

Prerequisite	An active Simulation file with NX Nastran as the specified solver environment
Toolbar	**Advanced Simulation® Edge-to-Edge Contact**
Simulation Navigator	Right-click **Simulation Objects® New Simulation Object® Edge-to-Edge Contact**

Temperature load enhancements for Nastran

What is it?

Note This topic is currently under construction

This release includes the following enhancements to the **Temperature Load** command in the NX Nastran and MSC Nastran solver environments:

- Temperature load support for 1D elements
- Temperature load enhancements for 2D elements

Temperature load support for 1D elements

You can use the new **Temperature on 1D Elements** option in the **Type** list to define temperature loads for CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements. Depending on the type of 1D element on which you're defining the load, you can specify:

- The temperature at both ends of the element.
- The temperature gradient at both ends of the element.

The new **Temperature on 1D Elements** capability corresponds to the Nastran `TEMPRB` bulk data entry.

Temperature load enhancements for 2D elements

You can use the new **Through Thickness (Temperature and Gradient)** and **Through Thickness (Top and Bottom)** options in the **Type** list to define the temperatures for 2D elements.

- Use the **Through Thickness (Temperature and Gradient)** option to define the temperature and gradient in the thickness direction for the element.
- Use the **Through Thickness (Top and Bottom)** option to define temperatures for additional membrane stress calculation at the lower and upper surface of 2D elements.

These new options correspond to the Nastran `TEMPP1` bulk data entry.

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® Temperature Load**
Simulation Navigator	Under the appropriate solution, right-click **Loads® New Load® Temperature Load**

Static acceleration load support (Nastran)

What is it?

Note This topic is currently under construction

In the NX Nastran and MSC Nastran environments, you can use the new **Acceleration** command to define a static acceleration load on your model. When you define an acceleration load, you must specify the magnitude of the load as well as the direction in which the load acts. The options in the **Acceleration** dialog box correspond to the fields on the `ACCEL` and `ACCEL1` bulk data entries.

Note The **Acceleration** constraint command that was available prior to NX 8.5 has been renamed to **Enforced Acceleration** for clarity.

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® Acceleration**
Simulation Navigator	In the appropriate solution or subcase, right-click **Loads® New Loads® Acceleration**

Support for bolt pre-loads on solid elements (NX Nastran)

What is it?

REVIEW NOTE

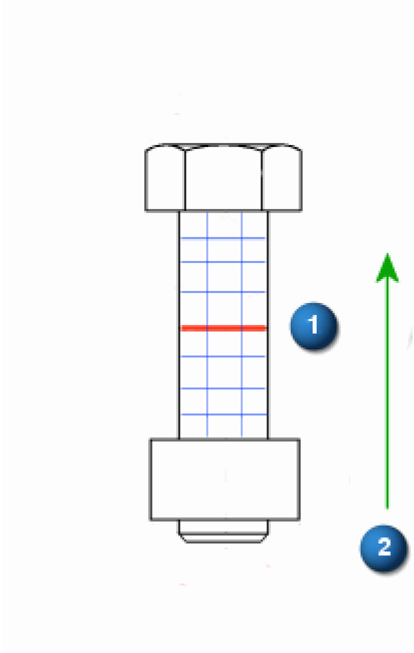
Issue: This topic is currently under construction for Beta

In the NX Nastran environment, you can now define a pre-load on a bolt that you model with solid elements. In the **Bolt Pre-Load** dialog box, you can use the new **Force on 3D Elements** option in the **Type** list to define the pre-loading force on solid CHEXA and CPENTA elements. In previous releases, you could only specify define a bolt pre-load on CBEAM and CBAR elements.

Note This capability is supported beginning in NX Nastran 8.5.

Defining a bolt pre-load on solid elements

With the **Force on 3D Elements** option, you must select a series of nodes that define a cut through the bolt at any location (1) on the interior of the bolt. You then specify a coordinate system and axis that NX Nastran uses to define the axis of the bolt (2).



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran as the specified solver
Toolbar	**Advanced Simulation® Bolt Pre-Load** 
Menu	**Insert® Mesh® Mesh Control**

Improvements to edge-based simulation regions (NX Nastran)

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

In the NX Nastran environment, you can now use the **Simulation Region** command to create a region comprised of discontinuous edges. Because you do not have to select continuous edges, edge-based **Simulation Regions** are now easier and more efficient to create.

The ability to create regions comprised of discontinuous edges utilizes the new `BEDGE` bulk data entry from the NX Nastran 8.5 release. The `BEDGE` entry offers several advantages over the `BLSEG` entry that was available in previous NX Nastran releases.

- With the `BEDGE` entry, you can select the edges in any order. With the `BLSEG` entry, edges must be selected in a continuous topological order.

- The `BEDGE` entry includes the element IDs along with a pair of grid (node) IDs to define a particular edge. The `BEDGE`

In the **Simulation Region** dialog box, you can now select which type of edge region you want to create:

- Select **Edge Region** from the **Type** list to define the region by selecting a series of polygon or element edges. These edges can be continuous or discontinuous. The **Edge Region** option corresponds to the new `BEDGE` bulk data entry.

- Select **Legacy Edge Region** from the **Type** list to define the region by selecting a series of continuous polygon or element edges. The **Legacy Edge Region** option corresponds to the legacy `BLSEG` bulk data entry.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran as the specified solver environment
Toolbar	**Advanced Simulation**® **Simulation Region** 
Simulation Navigator	Right-click **Simulation Objects**® **New Simulation Object**® **Edge-to-Edge Contact**

Contact support enhancements (ANSYS)

What is it?

Note This topic is currently under construction

This release includes significant enhancements to the contact modeling capabilities that are supported in the ANSYS solver environment. These enhancements include:

- Support for thermal contact
- Support for additional types of contact
- Enhancements to the **Simulation Region** capability for ANSYS models
- Element support enhancements

Thermal contact support

This release also includes a new **Thermal Contact** command that you can use to define thermal contact conduction between contacting surfaces in ANSYS thermal analyses. For more information, see New thermal boundary conditions (ANSYS).

Support for additional types of contact

In previous releases, NX only supported surface-to-surface contact for ANSYS models. In this release, NX now supports the following additional types of contact in both structural and thermal analyses:

- Node-to-surface contact, which uses a CONTA175 (node-to-surface contact element) to model flexible-flexible or rigid-flexible contact between a node and a 3D target surface (modeled with TARGE170 elements).
- Node-to-line contact, which uses a which uses a CONTA175 (node-to-surface contact element) to model flexible-flexible or rigid-flexible contact between a node and a 2D target surface (modeled with TARGE169 elements).
- Line-to-surface contact , which uses a CONTA177 (3D line-to-surface contact) element to model flexible-flexible or rigid-flexible contact between a 3D beam element and a surface or between the edge of a 2D element and a surface. TARGE170 elements are used to model the target surface.
- Line-to-line contact, which uses a CONTA172 (3D line-to-surface) element to model contact between beams undergoing large displacements.

Note Because NX now supports these different types of contact, the **Surface-to-Surface Contact** command in the ANSYS structural environment has been renamed to **Structural Contact** for clarity.

These new types of contact are available when you select the **Manual** option from the **Type** list in the **Structural Contact** and **Thermal Contact** dialog boxes. The software then uses the type of geometry you select in the **Simulation Region** dialog box for the source and target regions to determine the type of contact you are modeling.

For more information on these types of ANSYS contact, see the *ANSYS Mechanical APDL Contact Technology Guide*.

Simulation Region support for the ESURF command

You can now use the **Simulation Region** dialog box to control the use of the ANSYS `ESURF` command. In ANSYS, `ESURF` generates SURF151 or SURF152 elements that are overlaid on the free faces of selected existing elements.

When you are modeling contact with 2D elements, you can use the new **Correct direction of normal** option to ensure that the software generates target and contact elements that have normals in the correct direction.

Element support enhancements

PLANE182 and PLANE183 type elements are now available in the ANSYS structural environment to support line-to-line and node-to-line contact. In previous releases, these element types were only available in the ANSYS axisymmetric structural environment.

Where do I find it?

Structural Contact

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver and Structural as the specified analysis type
Toolbar	**Advanced Simulation® Structural Contact** 
Simulation Navigator	Under the active solution, right-click **Simulation Objects® New Simulation Object® Structural Contact**

Simulation Region

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver

Toolbar	**Advanced Simulation® Simulation Region**
Simulation Navigator	Right-click **Regions** ® **New Region**

New thermal boundary conditions (ANSYS)

What is it?

Note This topic is currently under construction

This release includes expanded support for ANSYS thermal boundary conditions. When you perform a thermal analysis in the ANSYS environment, you can now create:

- Thermal couplings.
- Support for thermal contact.
- Support for new thermal contact properties

Thermal couplings

You can use the new **Automatic Thermal Coupling** and **Manual Thermal Coupling** commands to create coupled degrees-of-freedom between nodes in a thermal analysis.

Thermal contact

You can use the new **Thermal Contact** command to define thermal contact conduction between contacting surfaces in ANSYS thermal analyses.

Support for new thermal contact properties

You can now define values for the following thermal contact properties in the **CONTA174 Real Constants** dialog box:

- **Thermal contact conductance** (TCC), which allows you to take into account the conductive heat transfer between the contact and target surfaces.
- The **Stefan-Boltzmann constant** (SBCT, which you can use to model radiative heat transfer. If you do not specify a value for the **Stefan-Boltzmann constant**, the radiation effect is excluded from the thermal contact calculations.
- The **Radiation view factor** (RDVF), which

Where do I find it?

Automatic Thermal Coupling and **Manual Thermal Coupling**

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver and Thermal as the specified analysis type
Toolbar	**Advanced Simulation® Automatic Thermal Coupling** or **Manual Thermal Coupling**
Menu	**Insert® Load® Automatic Thermal Coupling** or **Manual Thermal Coupling**

Thermal Contact

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver and Thermal as the specified analysis type
Toolbar	**Advanced Simulation® Thermal Contact**
Simulation Navigator	Under the active solution, right-click **Simulation Objects® New Simulation Object® Thermal Contact**

Constraining bolts to their pre-loaded length (Abaqus)

What is it?

Note This topic is currently under construction

You can use the new **Bolt Pre-Load Constraint** command to fix the length of the bolt after the pre-load has been applied.

This new command replaces the **Lock Bolts After Pre-Load Application** check box that was added to the **Solution** dialog box in the NX 8.0.1 release.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with Abaqus as the specified solver and Structural as the specified analysis type
Toolbar	**Advanced Simulation® Bolt Pre-Load Constraint**
Simulation Navigator	Under the active solution, right-click **Constraints® New Constraint® Bolt Pre-Load Constraint**

Heat Generation load improvements (Abaqus)

What is it?

REVIEW NOTE

Issue: This topic is currently under construction for Beta.

You can now use the **Heat Generation** command to define distributed heat fluxes on a selected:

- Polygon body.
- Polygon edge.
- Element.

In this release, for example, you can now use the **Heat Generation** command to define a body flux (flux per unit volume) on a selected polygon body. This corresponds to the Abaqus `*DFLUX` keyword with the `BF` parameter.

In previous releases, you could only define distributed heat fluxes on selected polygon faces or selected element faces. Because of this, you could only define a **Heat Generation** load on the faces of elements that were located on the external faces of the model. This meant that the **Heat Generation** load had very limited applicability.

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® Heat Generation** 
Simulation Navigator	Under the active solution, right-click **Loads® New Load® Heat Generation**

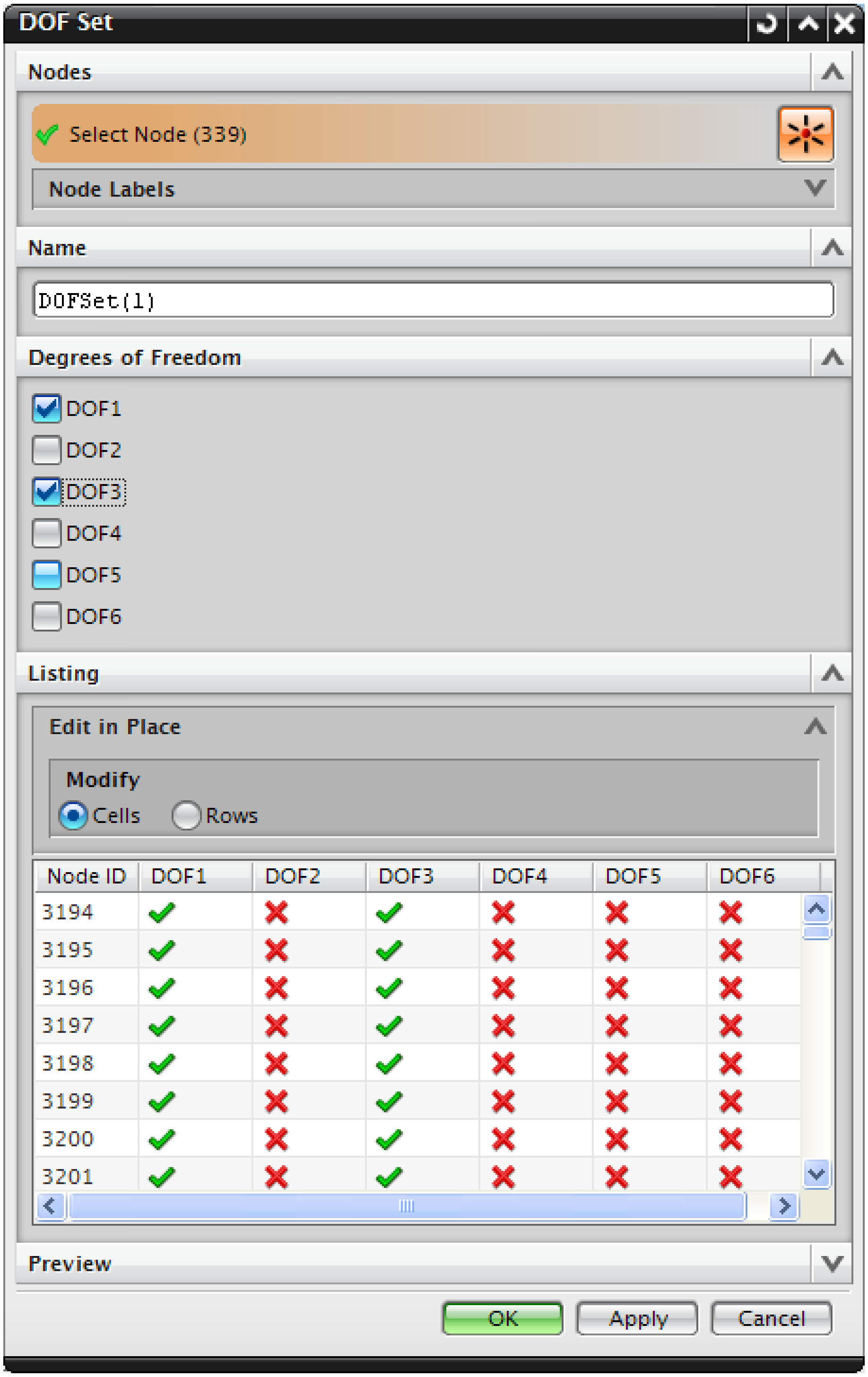
Enhancements to DOF Set command

What is it?

Note This topic is currently under construction.

In this release, the **DOF Set** command has been enhanced to make it easier to change the degrees of freedom settings for all nodes in the DOF set in a single action. This enhancement also improves the performance of defining the degrees of freedom for a large number of nodes. In previous releases, displaying the degrees of freedom in the **DOF Set** dialog box for a large number of nodes could be a slow process.

You can now select a large group of nodes and define the degrees of freedom for all of the selected nodes. If desired, you can still view and define the degrees of freedom in the **Listing** view in the **DOF Set** dialog box, as in previous releases.



Where do I find it?

Application	Advanced Simulation
Menu	**Insert® DOF Set® New**
Simulation Navigator	Right-click the **DOFSets** node® **New DOF Set**

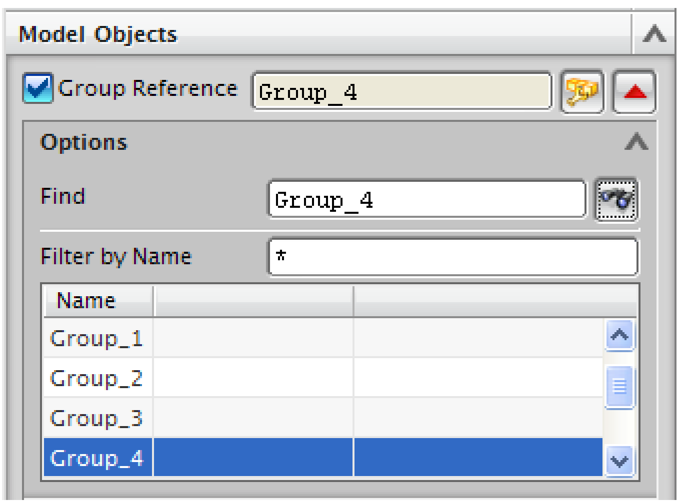
Find and filter reference objects

What is it?

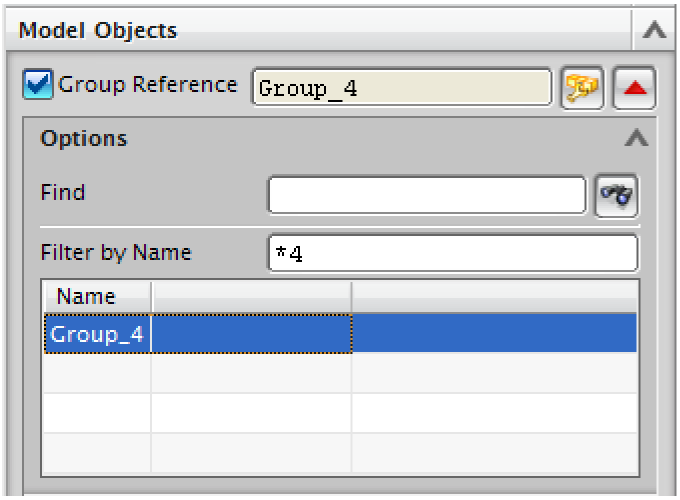
Note This topic is currently under construction.

Beginning with this release, all loads and boundary conditions dialog boxes (including solver-specific modeling objects, and some modeling objects) and physical property dialog boxes that have object reference blocks now have new **Find** and **Filter by Name** capabilities for locating reference objects. The **Filter by Name** command supports wildcards. The **Find** capability is case-sensitive.

For example, you can search for a reference group or filter the list of available reference groups.



Using the Find command to find Group_4



Using the Filter by Name command with a wildcard to find Group_4

Where do I find it?

Application	Advanced Simulation

Import, export, and solve enhancements

New advanced options for solver directory and file name

What is it?

REVIEW NOTE

Issue: This topic is under construction

In the Nastran, Abaqus, and ANSYS solver environments, you can use new options in the **Advanced Solver Options** dialog box to control:

- The name of the solver input file.
- The directory in which the solver input file is created.

In previous releases, the name and location of the solver input file was always based upon the name and location of the associated Simulation file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file active with NX Nastran, MSC Nastran, Abaqus, or ANSYS as the specified solver

Simulation Navigator	Right-click a solution® **Edit Advanced Solver Options** =

Ability to import simulation data from Gateway

What is it?

REVIEW NOTE

Issue: This topic is currently under construction for Beta.

You can now import simulation data directly from the Gateway application. In previous releases, you could only import simulation data from either the Advanced Simulation or Design Simulation applications.

Where do I find it?

Application	Gateway
Menu	**File® Import ® Simulation**

Nastran support enhancements

Subcase support improvements for certain modal solutions

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes improved support for the types of subcases you can create for certain Nastran modal solutions. These improvements include:

- Static subcase improvements
- Modal subcase improvements

Static subcase improvements

NX now provides better support for the Nastran STATSUB case control command. You can now create static subcases in SOL 103, 105, 107, 108, 109, 110, 111, and 112 in NX. Including static subcases in these types of solutions lets you form the differential stiffness for buckling analysis, normal modes, complex eigenvalue, frequency response, and transient response analysis.

Modal subcase improvements

You can now create modal subcases for normal modes data recovery in SOLs 110, 111, and 112.

- In SOL 110, yo can create a modal complex type of subcase.
- In SOL 111, you can create a modal frequency type of subcase.
- In SOL 112, you can create a modal transient type of subcase.

Where do I find it?

Application	Advanced Simulation
Prerequisite	In a Simulation file, a modal solution active with NX Nastran or MSC Nastran as the specified solver
Simulation Navigator	Right-click a the appropriate solution® **New subcase**
Menu	**Insert® Step-Subcase**

Nastran import and export support enhancements

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

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

Parameters

The following table details the changes in parameter support for this release.

Parameter	Description	Notes

System cell support updates

The following table details the changes in system cell support for this release.

System Cell	Name	Description	Notes

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**

Ability to preview solver syntax

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

In this release, you can now generate a preview of

Support for CBUSH1D elements and PBUSH1D properties

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

NX now supports Nastran CBUSH1D elements. The CBUSH1D element is a one dimensional spring and damper element that supports large displacements. You can use the CBUSH1D element to model rotational damping.

- The CBUSH1D element is a one dimensional version of the CBUSH element, without the rigid offsets.
- The CBUSH1D element outputs axial forces, relative axial displacements, and the relative axial velocity. It also poutputs stress and strain if you define stress and strain coefficients. You can use the options on the **Stress** tab in the **Structural Output Requests** dialog box to request this output.

CBUSH1D elements are useful, for example, in dynamic analyses. You can use CBUSH1D elements to model vibration control devices that have impedance values (stiffness and damping) that are frequency-dependent.

Creating CBUSH1D elements in NX

In Advanced Simulation, there are to ways to create a CBUSH1D element:

- You can use the **OD Mesh** dialog box to create a grounded CBUSH1D element at a particular point or on a particular node.
- You can use the **1D Mesh** dialog box to create CBUSH1D elements along a curve or face.

New PBUSH1D physical property table

NX also now supports the PBUSH1D bulk data entry. You can use the options in the new **PBUSH1D** physical property table dialog box to define the linear and nonlinear properties for CBUSH1D elements.

- You can define linear properties, including the element’s stiffness, viscous damping, and mass.
- You can define nonlinear properties, including:
  - o Geometric nonlinear functions of the axial forces versus the axial displacement.
  - o Geometric nonlinear functions of the axial forces versus the velocity.

- o Shock absorber characteristics which are defined by coefficients that describe the force versus velocity and displacement relationship.

Note Currently, NX only supports the `TYPE=EQUAT` option for the `PBUSH1D` bulk data entry.

Where do I find it?

0D Mesh

Application	Advanced Simulation
Prerequisite	An active FEM file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® 0D Mesh**
Menu	**Insert® Mesh® 0D Mesh**

1D Mesh

Application	Advanced Simulation
Prerequisite	An active FEM file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® 1D Mesh**
Menu	**Insert® Mesh® 1D Mesh**

PBUSH1D physical property table

Application	Advanced Simulation
Prerequisite	An active FEM file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® Physical Properties**
Menu	**Insert® Physical Properties**

Support for frequency dependent physical properties

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes support for the following frequency dependent Nastran physical properties:

- `PBUSHT`, which defines the frequency and stress dependent properties for a `PELAS` bulk data entry.

- `PDAMPT`, which defines the frequency dependent properties for a `PDAMP` bulk data entry.

- `PELAST`, which defines the frequency dependent properties for a `PELAS` bulk data entry.

In NX, new options have been added to the **PBUSH**, **PDAMP**, and **PELAS** physical property table dialog boxes to allow you to define frequency dependent properties. For example:

- In the **PBUSH** dialog box, you can use the new **Dependent Properties** optinos to specify fields that define frequency dependent stiffness values for the associated elements.

- In the **PDAMP** dialog box, you can use the new **Dependent Properties** options to specify fields that define values for viscous damping in extension and in rotation per radian for the associated elements.

In Nastran analyses, you can use frequency dependent properties in frequency response analyses (SOL 108 and 111).

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® Physical Properties** 
Menu	**Insert® Physical Properties**

New default to control shell thickness displays

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes the new **Shell Thickness Type Queries** customer default that you can use to control how NX evaluate the thickness of shell elements for:

- The **Plot Thickness Contours** command
- The **Thickness Information** command
- The **Element Thickness and Offset** option in the **Mesh Display** dialog box

Note This default only applies to 2D elements whose properties are defined with a **PSHELL** type of physical property table.

With the **Shell Thickness Type Queries** option, you can have the software evaluate the thickness of the element based on:

- The membrane thickness value, which corresponds to the `T` field on the `PSHELL` bulk data entry. This is the default method for evaluating the thickness of 2D elements.
- The fiber distances, which correspond to the `Z1` and `Z2` fields on the `PSHELL` bulk data entry. Fiber distances are used in the computation of stresses.

Where do I find it?

Application	Advanced Simulation
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Simulation**® **NASTRAN**® **General** tab

Import improvements for Nastran input files

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes a number of improvements for importing Nastran input files:

- Improved preservation of mesh content and naming when you import a Nastran input file that was originally created in NX.
- Support for relative path names in any `INCLUDE` files in the imported input file.
- New option to attach associated results to a new solution in Advanced Simulation.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file active with NX Nastran or MSC Nastran as the specified solver
Menu	**File® Import® Simulation**

Support for residual vector output results

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

In this release, you can now request the calculation of residual vectors from a modal analysis. Use the options on the new **Residual Vectors** tab in the **Structural Output Requests** dialog box to calculate residual vectors during the analysis. When you create a **Residual Vectors** output request, the software creates a `RESVEC` case control command in your Nastran input file.

Normal modes computed for use in a modal response solution (SOL 111 or SOL 112) are typically a reduced representation of a structure. The computed modes may represent most of the dynamic behavior of a structure, although higher frequency modes, which are not computed, can contribute to errors in the resulting response. Typically, the higher modes contribute statistically to the response. Requesting the calculation of residual vectors can improve the response by adding the missing static flexibility associated with these higher modes.

Note The `RESVEC` command is available beginning in the NX Nastran 8.0 release. It produces the same results that you could request with the `PARAM,RESVEC,YES` parameter prior to NX Nastran 8.0.

Where do I find it?

Residual Vector output request

Application	Advanced Simulation
Prerequisite	A Simulation file active with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects® Type** list **Structural Output Requests**
Menu	**Insert® Modeling Objects® Type** list **Structural Output Requests**

Residual Vector customer default

Application	Advanced Simulation
Prerequisite	A Simulation file active with NX Nastran or MSC Nastran as the specified solver
Toolbar	**File®** **Utilities®** **Customer Defaults** list **Structural Output Requests**
Location in Dialog Box	**Simulation®** **NASTRAN®** **Solution** tab

Response Simulation enhancements

What is it?

Note This topic is currently under construction.

This release includes two enhancements for Response Simulation:

- Analysis performance of PSD function responses in Random Response events has been improved.
- In the **New Event** dialog box, a **Duration Option** control is now available. For example, you can now set the event duration to equal the duration of the excitations.

Where do I find it?

Application	Advanced Simulation

Abaqus support enhancements

Abaqus keyword support enhancements

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

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

Keyword	Supported parameters	Import support	Export support	Notes

Orientation support for kinematic couplings

What is it?

In this release, when you create a **Kinematic Coupling** type of physical property table, you can use the new **Set Orientation** option to a local coordinate system and its orientation in which the constrained degrees-of-freedom are defined. You can specify a Cartesian, cylindrical, or a spherical coordinate system.

The new **Set Orientation** options correspond to the `ORIENTATION` parameter for the `*KINEMATIC COUPLING` keyword.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file active with Abaqus as the specified solver
Toolbar	**Advanced Simulation® Physical Properties** 
Menu	**Insert® Physical Properties**

Normalization parameter for frequency perturbation steps

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

In a **Frequency Perturbation** type of Abaqus solution step, you can use the new **Normalization Parameter** option to normalize the eigenvectors in the step.

- Select **Displacement** to normalize the eigenvectors to that the largest displacement or rotation in each vector is unity.

- Select **Mass** to normalize the eigenvectors with respect to the structure's mass matrix. With this option, the software scales the eigenvectors so that the generalize dmass for each vector is unity.

The **Normalization Parameter** corresponds to the `NORMALIZATION` parameter for the Abaqus `*FREQUENCY` keyword.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file active, with Abaqus as the specified solver, **General Analysis** as the solution type, and **Frequency Perturbation** as the active solution step
Simulation Navigator	Right-click the appropriate solution® **New Step**® **Frequency Perturbation**
Menu	**Insert**® **Step-Subcase**® **Frequency Perturbation**

New job execution controls

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes new options in the **Solver Parameters** dialog box that you can use to control how Abaqus executes the analysis. You can now specify options to:

- Manage memory allocation and disk usage.
- Specify high performance computing options, such as whether to use parallel processing and the number of processors to use.
- Control the precision of the nodal field output that Abaqus writes to the output database file (.odb).

For more information on controlling the execution of Abaqus analyses, see *Execution procedures* in the *Abaqus Analysis User's Manual*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	In a Simulation file with Abaqus as the specified solver, **General Analysis** as the solution type, and **Frequency Perturbation** as the active solution step
Simulation Navigator	Right-click the appropriate solution® **Solver Parameters**
Menu	**Analysis**® **Solve**® **Edit Solver Parameters**

Contact support enhancements

What is it?

This release includes several enhancements that expand the support for Abaqus contact analyses in NX. These enhancements include:

- Support for new constitutive models for softened contact.
- Support for additional contact solution controls.

Support for new constitutive models for softened contact

In the Abaqus environment, when you use the **Contact Pair** dialog box to define the behavior of the contacting surfaces as a **Softened contact relationship**, you can now use two new constitutive models control the motion of the surfaces in a mechanical contact analysis.

Advanced Simulation now supports both the **Tabular** and **Exponential** constitutive models for the pressure-overclosure relationship in Abaqus contact analyses:

- Select **Exponential** to define an exponential pressure-overclosure relationship. In an **Exponential** contact pressure-overclosure relationship, the surfaces begin to transmit contact pressure once the clearance between them, measured in the contact (normal) direction, reduces to c_0. The contact pressure transmitted between the surfaces then increases exponentially as the clearance continues to diminish.

  With the **Exponential** option, you specify the value of the contact pressure at zero clearance (p_0 and the clearance at which the contact pressure is zero (c_0).

- Select **Tabular** to define a piecewise linear pressure-overclosure relationship in a tabular form. In a **Tabular** relationship, the surfaces transmit contact pressure when the overclosure between them, measured in the contact (normal) direction, is greater than h_1, where h_1 is the overclosure at zero pressure.

  With the **Tabular** option, you specify the data pairs (h_1 and p_1) of pressure versus overclosure (where overclosure corresponds to negative clearance). You must specify the data as an increasing function of pressure and overclosure.

In previous releases, the only supported pressure-overclosure relationship was **Linear**, in which the surfaces transmit contact pressure when the overclosure between them, measured in the contact (normal) direction, is greater than zero.

Support for additional contact solution controls

Use the new **Contact Controls** modeling object to define additional solution controls for models that include contact between bodies. These additional controls allow you to control automatic stabilization of rigid body motions in contact problems that use viscous damping. For example, you can now control:

- The damping coefficient to use at the contact interface.
- The fraction of the damping that remains at the end of the step.
- The clearance at which the damping becomes zero.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file active with Abaqus as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects** 
Menu	**Insert® Modeling Objects**

Support for energy output

What is it?

This release includes new options on the **Energy** tab in the **Abaqus Structural Output Requests** dialog box that you can use to request the output of energy data. These new options correspond to the Abaqus `*ENERGY OUTPUT` keyword.

In the **ODB file output control** options, if you select the **Written into ODB file** check box and then select **History** from the **Filed/History Option** list, you can choose from the following new options:

- **ALLSD – Energy Dissipated by Automatic Stabilization**, which calculates the energy that is dissipated by automatic stablization for a selected group of elements or the entire model. This calculation includes both volumetric static stabilization and the automatic approach of contact pairs.
- **ALLSE – Recoverable Energy Strain**, which calculates the recoverable energy strain of a selected group of elements or the entire model.
- **ETOTAL – Total Energy Balance**, which calculates the total energy balance of the entire model.

These new energy output options are not available when you are performing:

- Eigenvalue buckling prediction.

- Natural frequency extraction.
- Complex eigenvalue extraction.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM or Simulation file active with Abaqus as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects**  **® Abaqus Structural Output Requests**
Menu	**Insert® Modeling Objects® Abaqus Structural Output Requests**

Input file processor data controls

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

When you solve an Abaqus solution, Abaqus creates a data file (`job-name.dat`) that contains information about the model's definition and a tabular output of results. The Abaqus analysis input file processor includes the model definition, the history definition, and messages identifying any error and warning conditions that were detected while processing the input data.

In this release, a new **Optional Controls** tab has been added to the **Solution** dialog box. You can use the options on that tab to control the amount of input file processor data that Abaqus writes to the data file. These new options correspond to the Abaqus `*PREPRINT` keyword. For example, you can control whether to print:

- Detailed information about the contact constraints generated by the contact pair definition data.
- A summary of model and history data.
- Information about input parameters and their values.

For more information, see *Output* in the *Abaqus Analysis User's Guide*.

Where do I find it?

Application	Advanced Simulation

Prerequisite	A Simulation file with Abaqus as the specified solver
Simulation Navigator	Right-click the Simulation® **New Solution**
Menu	**Insert® Solution**

Ability to format data as single or double precision

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

Use the new **Data Field Length** option in the **Advanced Solver Options** dialog box to control whether floating point nodal field data are written to the output database file (.odb) in single precision or double precision format. By default, data are written in single precision format.

- If you select **Small**, data are written in single precision format. For example:

```
**%==========================
**%       MODEL DATA
**%==========================
**%
*NODE, NSET=nset_csys0
          1, 9.500000E+02, 5.000000E+01, 5.000000E+01
          2, 9.500000E+02, 5.000000E+01, 1.000000E+02
          3, 9.500000E+02, 5.000000E+01, 1.500000E+02
          4, 9.500000E+02, 5.000000E+01, 2.000000E+02
```

- If you select **Large**, data are written in double precision format. For example:

```
**%==========================
**%       MODEL DATA
**%==========================
**%
*NODE, NSET=nset_csys0
          1, 9.5000000000000E+02, 5.0000000000000E+01, 5.0000000000000E+01
          2, 9.5000000000000E+02, 5.0000000000000E+01, 1.0000000000000E+02
          3, 9.5000000000000E+02, 5.0000000000000E+01, 1.5000000000000E+02
          4, 9.5000000000000E+02, 5.0000000000000E+01, 2.0000000000000E+02
```

In previous releases, you could not control the format in which nodal field data were written to the output database file.

Where do I find it?

Application	Advanced Simulation

Prerequisite	A Simulation file with Abaqus as the specified solver
Simulation Navigator	Right-click the solution® **Edit Advanced Solver Options**
Menu	**Analysis**® **Solve**® **Edit Advanced Solver Options**

Improved support for Abaqus materials

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes significant enhancements to the support for Abaqus materials in NX. These enhancements include:

- Support for orthotropic materials with temperature dependent shear strain/stress limits.
- Expanded support for gasket materials.
- A new option for defining activation energy in the hyperbolic sine creep law.
- New options for viscoelasticity material properties with a frequency domain.

Orthotropic materials with temperature dependent shear strain/stress limits

You can now use a field to define the **Shear** stress and strain limits for an Abaqus **Orthotropic** type of material. In previous releases, you could only define these limits as constant values. Temperature-dependent parameters for stress-based and strain-based failure measures (which correspond to the `*FAIL STRESS` and `*FAIL STRAIN` keywords) are now fully supported.

Expanded support for gasket materials

This release includes several enhancements to the support for **Gasket Behavior** materials in NX.

- When you create a **Gasket Behavior** material, you can now select **DAMAGE** from the **Type** list to use the damage elasticity model to define the gasket thickness-direction behavior. In previous releases, only the nonlinear elastic-plastic damage model was supported. You can use both temperature and non-temperature dependent pressure vs. closure data to define the loading and unloading curves.

When you create a **Gasket Behavior** material with the **DAMAGE** elasticity model, you can also define viscoelasticity properties in the frequency domain.

- When you create a **Gasket Behavior** and select the **ELASTIC-PLASTIC** option from the **Type** list, you can use the options on the **Creep** tab to include creep behavior.

Ability to define activation energy in the hyperbolic sine creep law

On the **Creep** tab, the **Kelvin-Maxwell (Rheological Constants (TYPE 111–222)** option and the **e*[sinh(f*SIGMA)]^g** option for the **Kelvin-Maxwell C3 Coefficient** together correspond to the Abaqus hyperbolic-sine law model for creep.

You can now use the new **DeltaH** option in the **Temperature Dependent Term Formulation** list to directly specify an activation energy value. When you select the **DeltaH** option, you can use the new **Delta H (per Mole)** to specify the activation energy.

In previous releases, the software derived the activation from the **Temperature Dependent Term Formulation** option as:

```
DeltaH = — LN[Temperature Dependent Term Formulation *R*T0]
```

Where *R* is the **Universal Gas Constant** (R=8.31434 joule per mole Kelvin) and *T0* is the **Reference Temperature (T0)**.

New options for viscoelasticity material properties with a frequency domain

When you define **Frequency Domain Viscoelasticity Properties**, you can use the new **Uniaxial and Volumetric** option in the **Preload** list to specify storage and loss moduli from both uniaxial and volumetric tests. In previous releases, you could only specify storage and loss moduli from either uniaxial or volumetric tests.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with Abaqus as the specified solver
Simulation Navigator	**Assign Materials** 
Menu	**Tools® Materials® Assign Materials**

Support for step options

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes new options on the **General** tab of the **Solution Step** dialog box that you can use to specify Abaqus step definition options. For example, you can now specify:

- The amplitude variation for loading magnitudes during the step.
- How to account for severe discontinuities, such as contact changes, during a nonlinear analysis.
- The extrapolation strategy to use in a nonlinear analysis.
- Whether Abaqus should use symmetric or unsymmetric matrix storage and solution.

These new options correspond to the Abaqus `*STEP` keyword.

Where do I find it?

Application	Advanced Simulation
Prerequisite	In a Simulation file with Abaqus as the specified solver
Simulation Navigator	Right-click the appropriate solution® **New Step**
Menu	**Insert**® **Step-Subcase**

Improved support for GAPUNI elements

What is it?

REVIEW NOTE

Issue: This topic is currently under construction for Beta.

This release includes improved support for Abaqus `GAPUNI` type elements. In the Abaqus environment in NX, when you use the **Contact Mesh** command to create point-to-point contact between two edges or a portion of two edges in your model, the software uses `*GAP` elements with the type `GAPUNI` to define that contact.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with Abaqus as the specified solver
Toolbar	**Advanced Simulation® Contact Mesh**
Simulation Navigator	Right-click the FEM file ® **New Mesh** ® **New Connection**

Support for 3D solid laminates

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

In this release, you can now model 3D solid, composite laminates in the Abaqus environment. There are two different ways to define 3D laminates in Advanced Simulation:

- You can select the new **Basic Solid Laminate** physical property table option to define the properties of the laminate. For example, you can specify the stacking direction with respect to a pair of element faces. You can also define the material, thickness, and orientation angle for each layer of plies.

- You can select the new **Solid Laminate** physical property table to use the **Solid Laminate Modeler** dialog box to define the layups' global plies.

In previous releases, you could only define 2D laminate composites in the Abaqus environment.

Where do I find it?

Application	Advanced Simulation
Prerequisite	In a Simulation file with Abaqus as the specified solver
Toolbar	**Advanced Simulation® Physical Properties**
Menu	**Insert® Physical Properties**

ANSYS support enhancements

ANSYS command support enhancements

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes import and export enhancements for the following ANSYS commands:

Element support enhancements

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

This release includes support for the following types of ANSYS elements:

ANSYS element type	Description	Supported analysis environments	Command to create the element in NX	Notes
`LINK34`	1D, 2 node element	Thermal Axisymmetric Thermal	**1D Mesh** 	Use the new **LINK34** physical property table dialog box to define the convection surface area of the element. • You can specify the convection surface area. • You can have the software compute

ANSYS element type	Description	Supported analysis environments	Command to create the element in NX	Notes
				the area of the element based on the total surface area of the underling elements.
CONTA178	1D, 2 node element	Structural Axisymmetric Structural	**1D Mesh**	
PIPE228	1D, 2 node element with an orientation node	Structural	**1D Mesh**	
SHELL131	2D, 8 or 6 node parabolic element (quadrilateral or triangular)	Thermal	**2D Mesh**	
SHELL132	2D, 4 or 3 node linear element (quadrilateral or triangular)	Thermal	**2D Mesh**	
SOLID226	3D, 15 or 20 node element (Hexahedral 20, Pyramid 15, and Wedge 15)	Structural Thermal	**3D Swept Mesh**	
SOLID227	3D, 10 node element	Structural Thermal	**3D Tetrahedral Mesh**	

New option for formatting ANSYS input files

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

Use the new **Do not use block format for nodes and elements** formatting option in the **Solver Parameters** dialog box to control whether NX uses the `NBLOCK` and `EBLOCK` commands to write out the ANSYS input file.

- Select the **Do not use block format for nodes and elements** check box if you want NX to write out the ANSYS input file without using the `NBLOCK` and `EBLOCK` commands.

- Clear the **Do not use block format for nodes and elements** check box if you want NX to use the `NBLOCK` and `EBLOCK` commands to write out the ANSYS input file.

Writing out node and element data to the ANSYS input file in the *blocked* format can greatly reduce the amount of time ANSYS needs to read the file. This difference is most noticeable with very large models.

Where do I find it?

Application	Advanced Simulation
Prerequisite	In a Simulation file with ANSYS as the specified solver
Simulation Navigator	Right-click the appropriate solution® **Edit Solver Parameters**
Menu	**Analysis**® **Solve**® **Edit Solver Parameters**

Ability to apply a flame temperature to SURF152 elements

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

Use the new **Use radiation to apply flame temperature on SURF152 exra node** option in the **Solution** dialog box to define the radiation effect of a flame on a surface, given the temperature of the flame and the emissivity of the surface.

When you select this option, the software changes how it handles the properties that you have defined for a **Radiation** load in your solution.

- The specified **Ambient Temperature** value becomes the temperature for the flame.

- The specified **Surface Emissivity** value becomes the emissivity of the surface to which the flame is applied.

When you export or solve your model:

- The software creates SURF152 elements in your ANSYS input file. These SURF152 elements are overlaid on the element faces you selected in the **Radiation** dialog box. Each SURF152 element as an extra node at the center of each element's face where ANSYS applies the specified **Ambient Temperature** as the flame temperature.

- The software writes out the specified **Surface Emissivity** value out as an emissivity property for the material applied to the SURF152 elements.

The advantage of using SURF152 elements to apply the flame radiation is that you can apply other thermal boundary conditions, such as a **Heat Flux** or **Convection**, to the same underlying surface.

Only a single flame radiation value is supported in a given solution

You cannot apply multiple flame radiation definitions to the same surface. If your solution or subcase includes multiple **Radiation** definitions and you select the **Use radiation to apply flame temperature on SURF152 exra node** option, software applies the last **Radiation** definition you created during the analysis.

Where do I find it?

Use radiation to apply flame temperature on SURF152 exra node option

Application	Advanced Simulation
Prerequisite	In a Simulation file with ANSYS as the specified solver and Thermal as the specified analysis type
Simulation Navigator	Right-click the Simulation® **New Solution**
Menu	**Insert**® **Solution**

Radiation load

Application	Advanced Simulation
Prerequisite	In a Simulation file with ANSYS as the specified solver and Thermal as the specified analysis type

Toolbar	**Advanced Simulation® Radiation**
Menu	Right-click **Loads** ® **New Load** ® **Radiation**

Display support for 2D element thickness

What is it?

REVIEW NOTE

Issue: This topic is currently under construction

You can now use the **Element Thickness and Offset** option in the **Mesh Display** dialog box to display the thickness of the 2D elements in an ANSYS axisymmetric thermal model. In previous releases, this option was not supported in the ANSYS axisymmetric thermal environment.

You can create a display of the element thicknesses to verify that the thickness values are defined correctly for the model. An element thickness display can also help you locate areas in your model in which the 2D elements have no defined thickness values.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file active with ANSYS as the specified solver and **Axisymmetric Thermal** as the analysis type
Menu	**Preferences® Mesh Display**

Post Processing

Annotation

What is it?

Note This topic is currently under construction.

You can now create, edit, and manage persistent annotations for post-processing displays. This feature replaces the min/max markers found in earlier releases.

Annotations you create may be:

- Attached to specified nodes or elements in the model.

- Attached to nodes that meet specified results criteria. When you change the specified results in the post view, these annotations update to reflect the new node or element locations and results criteria.

- Unattached annotations that apply to the entire model.

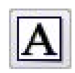
Annotations can contain any text you specify, and may optionally include entity IDs and/or the results value of the attached node or element.

Annotations are listed in the **Post Processing Navigator**, under the **Post View** node. You can hide and show, rename, edit, and delete annotations.

When you create a new post view, the software automatically creates two new annotations, **Minimum** and **Maximum**. These annotations duplicate the min/max markers found in previous releases. To display the default annotations, in the **Post Processing Navigator**, expand the **Post View** node, expand the **Annotations** node, and select the **Minimum** or **Maximum** visibility check box.

Why should I use it?

Where do I find it?

Application	Advanced Simulation
Prerequisite	A post view of loaded results.
Toolbar	**Post Processing** toolbar® **New Annotation**  **Post Processing** toolbar® **Drag Annotation** 
Menu	**Tools**® **Results**® **New Annotation** **Tools**® **Results**® **Drag Annotation**

Results Manipulation enhancements

What is it?

Note This topic is currently under construction.

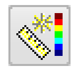
The **Results Manipulation** command has been enhanced to include two new manipulation types:

- The **Reduction** type enables you to manipulate multiple components of a result type using NX expressions. For example, you could use this to perform a scalar reduction of tensor data using your own formulation.

- The **Transient Reduction** type enables you to manipulate multiple components of a result type across multiple time steps or iterations.

In addition, all **Results Manipulation** types have been enhanced to enable you to assign units to results and provide additional options for associating manipulated results with Simulations and solutions.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Results loaded in the **Post Processing Navigator**.
Toolbar	**Post Processing® Results Manipulation** 
Menu	**Tools® Results® Results Manipulation**

Streamline seed set enhancements

What is it?

Note This topic is currently under construction.

The ability to quickly generate seed sets for streamline displays of velocity results has been enhanced with the addition of four new **Pick** methods based on nodal locations:

- **All on Face** creates a seed point at each node on a selected face.
- **All on Edge** creates a seed point at each node on a selected edge.
- **Sample on Face** creates a seed set by sampling node locations on a face that are farther apart than the specified **Minimum Distance**.
- **Sample on Edge** creates a seed set by sampling node locations on an edge that are farther apart than the specified **Minimum Distance**.

Why should I use it?

Use these methods to create representative seed seeds more quickly and consistently.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A results set loaded that contains velocity results, and a post view of velocity results.

Toolbar	**Post Processing® Edit Post View** . From the **Color Display** list, choose **Streamlines**, and **Options** (**Streamline Parameters**). In the **Streamline Parameters** dialog, click **Create**.

Custom post view templates

What is it?

Note This topic is currently under construction.

You can now edit post view templates customized, partial post view templates.

Partial templates are used to apply a limited set of display options to the current post view, without overriding other settings. For example, you could create a template that assigns specified edge, face, and contour settings, without overriding the current results or color bar settings.

You can edit using any XML editor and the schema file provided. You can specify the XML editor executable using the `UGII_CAE_POST_TEMPLATE_EDITOR` environment variable, or by right-clicking the **Templates** node in the **Post Processing Navigator** and choosing **Editor Option**.

Why should I use it?

Partial templates save time and ensure consistency when creating complicated or customized post views, such custom color bars and legends.

Using partial templates, you can create a library of stored post view settings. If you store the template .xml files in a network directory specified by the `UGII_CAE_POST_TEMPLATE_USER_DIR` environment variable, all users on the network can access the same template library, ensuring consistent displays across the enterprise.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved CAE model, results loaded, and a default post view.
Post Processing Navigator	Right-click the **Templates** node® **Editor Option** Right-click a template® **Edit**

Explicitly associate results with solutions

What is it?

Note This topic is currently under construction.

Beginning with this release, you can explicitly associate a solver results with a solution in your CAE model.

In previous releases, and currently by default, the associated results file was inferred, assuming that the solver file was located in the same directory as the Simulation file, and that the solver results file name followed standard naming conventions (*simulation_name-solution_name.**, where the file extension is .op2, .bun, .unv, and so on).

Using this feature, you can:

- Explicitly specify a solver results file by name.
- Specify a directory other than the directory where the simulation file is located. NX will then infer the result based on file-naming conventions.
- Specify alternate units for results.

Why should I use it?

Explicitly specify associated results files when solver output is not stored in the same directory as the Simulation file. For example, you may archive solver output on a network server. You may also need to explicitly associate results with a solution if the solver output file name does not follow standard naming conventions for NX.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved CAE model.
Simulation Navigator	Right-click the **Results** node® **Specify**

Miscellaneous post processing enhancements

What is it?

Note This topic is currently under construction.

- Enhancements to results imported into a Simulation.
- Ability to display imported results on Simulation groups.

- Ability to display results on groups when more than one Simulation is loaded.

Why should I use it?

Where do I find it?

Application	Advanced Simulation
Prerequisite	
Toolbar	
Menu	® ®

Optimization

Tosca Shape Optimization

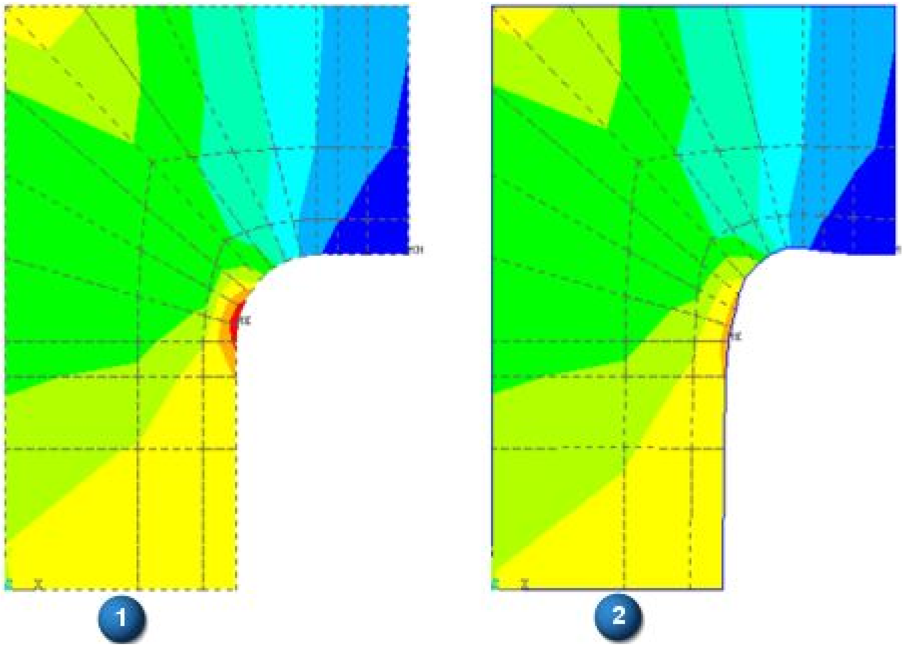
What is it?

Note This topic is currently under construction.

You can now define a shape optimization using the Tosca Shape Optimization solver by FE Design. Shape optimization suggests specific detailed improvements to an existing design by deforming the finite element mesh. The optimization algorithm arrives at the optimum contour of the component while taking all boundary conditions into consideration. You typically use shape optimization at the end of the design process, when the boundary conditions, material selection, and manufacturing method are defined, and only minor changes and improvements are allowed.

The objective of a shape optimization can be to minimize stress concentrations or maximize selected natural frequencies, under defined constraints such as volume, manufacturing restrictions, and symmetry. To achieve the objective, the optimization algorithm displaces individual nodes that you specify as design nodes.

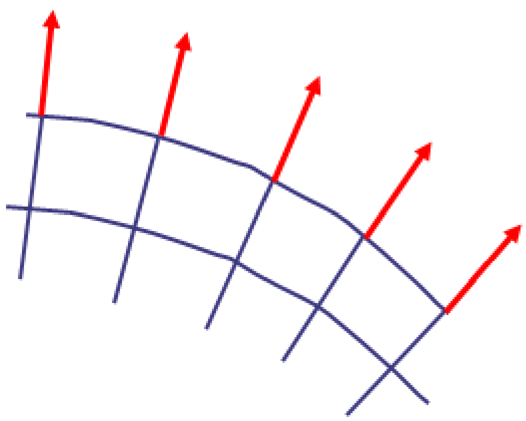
Each design node is displaced with the objective of reducing the local stresses. In the high-stressed area, the design nodes are displaced outward and the structure grows; in the low-stressed area, the design nodes are displaced inward and the structure shrinks.



(1) Starting design; (2) Optimized design; notice the reduced notch stress

The design variables are the positions of the design nodes and the amounts of displacement that the algorithm applies to these design nodes. The displacement direction is determined based on the FE polygon geometry:

- For 2D meshes, this is the element edge normal.
- For 3D meshes, this is the element surface normal.



2D mesh design nodes, displacement direction

Each design cycle begins with an FE analysis performed by NX Nastran, which solves for the stress results in the component. Based on the results, the optimization modifies the positions of the design nodes on the surface of the component, taking the local stresses into consideration. Each design node is independent of its neighboring node. The optimization homogenizes the stresses by minimizing the deviation from an automatically determined reference stress value. Then, NX Nastran solves the model again, using the updated mesh, for the new stress results. These iterations repeat until the specified maximum number of iterations is reached.

The output of the optimization is an NX Nastran bulk data file that contains the modified mesh.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A SOL101 static solution or SOL103 modal solution
Menu	**Insert**® **Shape Optimization**
Simulation Navigator	Right-click the Simulation file® **New Solution Process**® **Shape Optimization**

Geometry Optimization enhancements

What is it?

Note This topic is currently under construction.

This release expands the set of response types you can choose from for the Geometry Optimization design objective and constraints. You can now define:

- Displacement
- Stress
- Strain
- Temperature

These response types are supported using the **Result Measure** command that was introduced in NX 8.0. To fully support optimization, the **Result Measure** command has been enhanced to return the absolute maximum, minimum, and mean average.

Where do I find it?

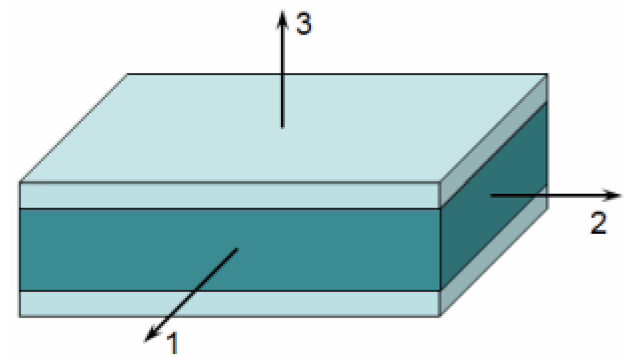
Application	Advanced Simulation, Design Simulation
Menu	**Insert**® **Geometry Optimization**
Simulation Navigator	Right-click the Simulation file® **New Solution Process**® **Geometry Optimization**

NX Laminate Composites

Core ply material

What is it?

Use the new **Core** ply material to define the core ply material and strength correction factor when you model failure indices for laminates with a core ply.



3-ply laminate with a core ply in the middle

The core ply material can be either isotropic or orthotropic.

The strength correction factor, K, is used to calculate the failure index as follows:

$$F_{13} = t_{13}/(S_{13}K)$$

$$F_{23} = t_{23}/(S_{23}K)$$

where:

- S_{13} and S_{23} are the shear allowable stresses.
- t_{13} and t_{23} are the core ply shear stresses in ply coordinate system.
- F_{13} and F_{23} are the core ply shear stress failure index in both shear directions.
- K is the strength correction factor that can be either constant or a function of the ply thickness.

You specify the strength correction factor in the **Laminate Ply Material** dialog box.

Why should I use it?

The core ply material correctly models laminate core failure for honeycombed laminates.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Toolbar	**Laminates® Ply Materials**
Menu	**Insert® Laminate® Ply Materials**
Location in dialog box	**Type** list® **Core® Create**

Core strength results

What is it?

REVIEW NOTE

Issue: This topic is under construction

Interlaminar failure theory

What is it?

Use the new **Use Material Shear Allowables** option to compute interlaminar transverse shear failure results using the shear stress allowable that you define in the **Shear (SS)** box in **Strength** tab of the **Isotropic Material** dialog box.

In previous releases, you could only compute interlaminar transverse shear failure results using the value that you set for the whole laminate in the **Allowable Stress For Bonding** box. To use this method in NX 8.5, from the **Interlaminar Failure** list, select **Use Allowable Stress For Bonding**.

Why should I use it?

This new interlaminar failure theory lets you use material property values.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Toolbar	**Laminate® Laminate Physical Property** or **Solid Laminate Physical Property**

Menu	**Insert**® **Laminate**® **Physical Property**® **Physical Property** or **Solid Physical Property**
Location in dialog box	**Laminate Properties** group® **Interlaminar Failure** list® **Use Material Shear Allowables**

Import FiberSIM materials

What is it?

REVIEW NOTE

Issue: This topic is under construction

When you import layups from FiberSIM using the HD5 format, you can now import FiberSIM materials or override existing FiberSIM materials in NX.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM. hD5 import
Toolbar	**Laminate**® **Import Global Layup** 
Menu	**Insert**® **Laminate**® **Global Layup**® **Import Layup**
Location in dialog box	**Import Materials** or **Override Existing Materials**

Ply extensions

What is it?

REVIEW NOTE

Issue: This topic is under construction

Single Ply per 3D Layer option

What is it?

REVIEW NOTE

Issue: This topic is under construction

Use the new **Single Ply per 3D Layer** option to control the number of 3D layers in a 3D mesh that is created when you inflate your laminate.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Toolbar	**Laminate**® **Extrude Laminate** or **Fill Laminate**
Menu	**Insert**® **Laminate**® **Inflation**® **Extrude** or **Fill**
Location in dialog box	**Settings** group® **Single Ply per 3D Layer**

3D inflation for Abaqus

What is it?

REVIEW NOTE

Issue: This topic is under construction

The **Extrude Laminate** and **Fill Laminate** commands are supported in the Abaqus solver environment.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Toolbar	**Laminate**® **Extrude Laminate** or **Fill Laminate**
Menu	**Insert**® **Laminate**® **Inflation**® **Extrude** or **Fill**

Linear elements supported for inflation

What is it?

REVIEW NOTE

Issue: This topic is under construction

You can now inflate 2D laminates that are meshed with 2D linear elements.

In previous releases, you could only inflate 2D laminates that were meshed with 2D quadratic elements.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Toolbar	**Laminate®** **Extrude Laminate** or **Fill Laminate**
Menu	**Insert®** **Laminate®** **Inflation®** **Extrude** or **Fill**

Copy 2D laminate to 3D laminate and back

What is it?

You can now easily create a **Solid Laminate** physical property from a **Laminate** physical property and vice-versa.

The copied physical property has the same ply layup and stacking recipe as his 2D or 3D counterpart.

The ply failure theory from a 2D laminate becomes the ply failure theory for each ply in the 3D laminate.

If the 3D laminate that you copy has multiple ply failure theories defined, the ply theory that is referenced by most 3D laminate plies is assigned to the 2D laminate.

Why should I use it?

The **Copy as Solid Laminate** and **Copy as Shell Laminate** commands provide more flexibility when you create 2D and 3D laminates from existing laminates.

Where do I find it?

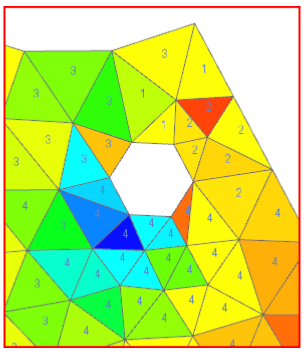
Application	Advanced Simulation
Prerequisite	You must work in the FEM.
Toolbar	**Advanced Simulation®** **Physical Properties**
Menu	**Insert®** **Physical Properties**
Location in dialog box	**Selection** group® right-click a **Laminate** physical property® **Copy as Solid Laminate** **Selection** group® right-click a **Solid Laminate** physical property® **Copy as Shell Laminate**

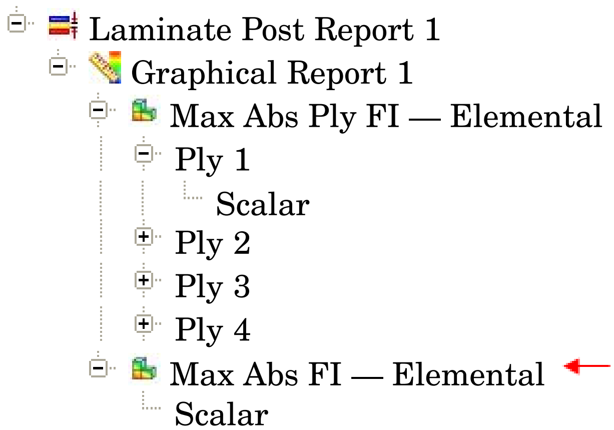
Show Critical Ply ID

What is it?

Use the new **Show Critical Ply ID** option to display the ply IDs over the laminate graphical report results of envelopes for the critical ply. The envelopes show the results for all the selected solutions, sub cases, and iterations at each element, for the critical ply and each ply separately. Critical ply IDs are only available for the critical ply. They are stored in the result nodes that do not contain the word Ply; for example, **Max Abs FI — Elemental** and **Max Abs Bond FI — Elemental**.

Example The graphic displays the **Max Abs FI — Elemental** results with the critical ply IDs on the elements. The ply IDs range from 1 to 4 and represent the ply which has the maximum **Max Abs Ply FI — Elemental** result.





Why should I use it?

The **Show Critical Ply ID** option lets you identify visually which ply is critical on the model.

Where do I find it?

Application	Advanced Simulation
Prerequisite	The graphical report result sets must include envelope result sets.
Post Processing Navigator	Select an enveloping result set® right-click the **Post View** node® **Show Critical Ply ID**

Ply location option for shell stress resultant

What is it?

When you create a laminate spreadsheet report or laminate graphical report, use the **Ply Stress Calculation Location** option to specify one or more calculation locations for ply results. The ply results can be computed at the following locations:

- **Bottom**
- **Middle**
- **Top**

In previous releases, for the laminate spreadsheet report, you could only request ply results at the top, middle, or bottom of the ply or at all three locations. For the laminate graphical report, the ply stress results were calculated only at the middle of the plies.

Why should I use it?

Use the **Ply Stress Calculation Location** option to select any combination of ply stress calculation location for the shell stress resultants of the laminate spreadsheet and graphical reports.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the Simulation file and a laminate post report must be active. You must select **Solver Shell Stress Resultants** on the **Options** tab in the **Spreadsheet Report** or **Graphical Post Report** dialog box.
Toolbar	**Laminate® Spreadsheet Report** or **Graphical Post Report**
Menu	**Insert® Laminate® Advanced Post Reporting® Spreadsheet Report** or **Graphical Post Report**
Simulation Navigator	Right-click the laminate post report node® **Create Spreadsheet Report** or **Create Graphical Post Report**
Location in dialog box	**Options** tab® **Input Selection** group® **Ply Stress Calculation Location**

Enveloping rules in the spreadsheet report

What is it?

When you create a laminate spreadsheet report, you can now specify the enveloping rule for each ply result. The following enveloping rules are available:

- **Max & Min**
- **Min**
- **Max**
- **Max Abs.**
- **Min Abs.**

In the previous release, enveloping rules were hardcoded for the output results.

- Ply stress, ply strain, and failure index results always displayed the maximum absolute value over all plies and subcases.
- Strength ratio results always displayed the minimum absolute value over all plies and subcases.
- Margin of safety results always displayed the minimum value over all plies and subcases.

Why should I use it?

This enhancement gives you more control on the rules used to envelope your results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the Simulation file and a laminate post report must be active.
Toolbar	**Laminate® Spreadsheet Report**
Menu	**Insert® Laminate® Advanced Post Reporting® Spreadsheet Report**
Simulation Navigator	Right-click the laminate post report node® **Create Spreadsheet Report**
Location in dialog box	**Output** tab

Minimum absolute enveloping rule

What is it?

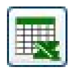
When you create a laminate spreadsheet report or laminate graphical report, you can now select the minimum absolute enveloping rule, **Min Abs.**, for each output result type.

Why should I use it?

Use the minimum absolute enveloping rule to identify results, negative or positive, that are closest to zero.

Example The minimum absolute enveloping rule can be useful when the strength ratio is negative on one ply and positive on another ply, such as when the strength ratio comes from a quadratic failure theory.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must work in the Simulation file and a laminate post report must be active.
Toolbar	**Laminate® Spreadsheet Report**  or **Graphical Post Report** 
Menu	**Insert® Laminate® Advanced Post Reporting® Spreadsheet Report** or **Graphical Post Report**
Simulation Navigator	Right-click the laminate post report node® **Create Spreadsheet Report** or **Create Graphical Post Report**
Location in dialog box	**Output** tab in the **Spreadsheet Report** dialog box **Options** tab in the **Graphical Report** dialog box

Changes to the laminate menu and toolbar

What is it?

The following new command is available on the **Laminate** menu and toolbar in the FEM:

Solid Laminate Physical Property  — Use this command to open the **Solid Laminate Modeler** dialog box.

In the previous release, you could access the **Solid Laminate Modeler** dialog box only from the **Physical Property Tables Manager** dialog box.

The following post processing commands are now available on the **Laminate** menu and toolbar in the Simulation file:

Spreadsheet Report — Use this command to open the **Spreadsheet Report** dialog box.

Graphical Post Report — Use this command to open the **Graphical Report** dialog box.

Quick Report — Use this command to open the **Quick Report** dialog box.

In the previous release, you could access these post processing commands only from the laminate post report node in the **Simulation Navigator**.

Where do I find it?

Solid Laminate Physical Property command

Application	Advanced Simulation
Prerequisite	You must work in the FEM within the Nastran or Abaqus solver environment.
Toolbar	**Laminate® Solid Laminate Physical Property**
Menu	**Insert® Laminate® Physical Property® Solid Laminate Physical Property**

Post processing commands

Application	Advanced Simulation
Prerequisite	You must work in the Simulation file and a laminate post report must be active.
Toolbar	**Laminate® Spreadsheet Report** or **Graphical Post Report** or **Quick Report**
Menu	**Insert® Laminate® Advanced Post Reporting® Spreadsheet Report** or **Graphical Post Report** or **Quick Report**
Simulation Navigator	Right-click the laminate post report node® **Create Spreadsheet Report** or **Create Graphical Post Report** or **Create Quick Report**

Durability

Random durability event

What is it?

Use the new **Random Durability Event** to compute life and damage for structures that are subjected to random excitations. The excitations are defined by a Response Simulation solution process.

In the **Random Durability Event** dialog box, you:

- Specify a Response Simulation solution process and its random event.
- Define a random fatigue durability object that contains random fatigue settings and fatigue life output requests.
- Define a solve options durability object that contains element and material overrides.

With the Von Mises power spectral density (PSD) for an element-node and stress life material properties as inputs, the durability solver computes fatigue life and damage using either the narrow-band (Miles) approach or the wide-band (Dirlik) approach.

Why should I use it?

The random durability event is used to predict life and damage for structures that are subject to random loads that are characterized by a power spectral density.

Where do I find it?

Application	Advanced Simulation
Prerequisite	NX Response Simulation Random Event
Toolbar	**Durability® Random Durability Event**
Menu	**Insert® Durability® Event® Random**
Simulation Navigator	Right-click the Durability solution process node® **New Event® Random**

Specified S-N and E-N curves

What is it?

You can now define **Stress-Life** (S-N) and **Strain-Life** (E-N) curves that the durability solver uses to calculate fatigue life and damage results.

When you define S-N and E-N curves, rather than coefficients and exponents, the durability solver treats them differently depending on the specified fatigue life and the durability event.

For static and transient events, if you define:

- **Stress Life** as the fatigue life criterion, the durability solver uses the specified S-N curve.

- **Strain Life Maximum Principal** or **Strain Life Maximum Shear** as the fatigue life criterion, the durability solver uses the specified E-N curve.

- **Smith Watson Topper** as the fatigue life criterion, the durability solver does not process the selected elements and creates a *[durability solution process name]-[event name]_InvalidMaterialPropertiesGroup_Fatigue_Life* diagnostic group.

- **BWI** or **TWI**, as the fatigue life criterion, the durability solver ignores the defined S-N and E-N curves and uses the settings defined under **Weld Data** in the **Fatigue** durability object.

For a random event, the durability solver uses the least square method to fit the specified S-N curve and find coefficients. The coefficients are then used in the equations for calculating fatigue life and damage results from the random event.

Why should I use it?

Specifying measured S-N and E-N curves instead of approximating them using coefficients ensures more accuracy in computing durability parameters.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation® Assign Materials** or **Manage Materials**
Menu	**Tools® Materials® Assign Materials** or **Manage Materials**

Location in dialog box	**New Material** subgroup® **Type** list=**Isotropic**® **Create material** ® **Isotropic Material** dialog box® **Durability** tab (S-N curve)**Fatigue Strength** group® **Stress-Life Data** list ® **Field** (E-N curve) **Fatigue Ductility** group® **Strain-Life Data** list ® **Field**

Durability objects

What is it?

Use the durability objects to define durability solver settings. You can define the following durability objects:

- **Strength** durability object that contains a stress criterion, stress type, and strength output requests.

- **Fatigue** durability object that contains fatigue life settings, fatigue life output requests, fatigue safety factor settings, and fatigue safety factor output requests.

- **Random Fatigue** durability object that contains random fatigue settings and fatigue life output requests.

- **Axis Search** durability object that contains stress axis method and settings.

- **Solve Options** durability object that contains element and material overrides.

The following table lists which durability objects must be defined for which durability event or damage evaluation. You can only define one object of each type per event or damage evaluation.

	Strength	**Fatigue**	**Random Fatigue**	**Axis Search**	**Solve Options**
Static Durability Event	X	X		X	X
Transient Durability Event	X	X		X	X

	Strength	Fatigue	Random Fatigue	Axis Search	Solve Options
Random Durability Event			X		X
Evaluate Damage		X			

Your Simulation file can contain multiple durability objects of each type.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Durability® Manage Durability Objects** 
Menu	**Insert® Durability® Manage Objects**

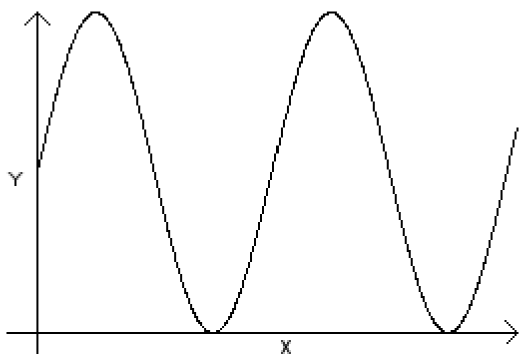
Function excitation for a static event

What is it?

Use the new **Function** excitation for a static durability event to create an excitation by multiplying an existing static load case with a general AFU table function that you define.

In the **Function** dialog box, you:

- Select a subcase and a general function.
- Define a scale and an offset.



To create the general AFU table function, you use the **XY Function Manager** and **XY Function Editor** commands.

The AFU function that you create must have the following properties:

- The **Purpose** and **Function Type** lists must be set to **General**.

- The abscissa data type must be time that is evenly spaced.
- The ordinate data type can be one of the following: **Unknown**, **Unitless Scalar**, **Unitless Real**, and **Unitless Integer**.

Note A given static event can have multiple excitations of the same type. The available types are now:

- Load patterns
- Result paths
- General functions

Why should I use it?

This command enables the creation of quasi-static scaling functions of a load case. It is useful when inertial and dissipative effects can be ignored in the transient response of the structure.

Where do I find it?

Application	Advanced Simulation
Prerequisite	In the **Static Durability Event** dialog box, the **Excitation Type** list must be set to **Function**.
Simulation Navigator	Right-click a static event node® **New Excitation**® **Function** dialog box

Flexible multibody durability analysis

What is it?

You can now perform a durability analysis on results from the multibody dynamics solver stored in a **.mdf* file in conjunction to the results from the corresponding NX Nastran SEMODES 103 — Flexible Body solution stored in a **.op2* file.

For each element node, the durability solver recovers stress and strain histories by scaling the modal displacement histories from the MDF file with mode shapes from the OP2 file. The computed stress and strain histories are then used by the durability solver to calculate fatigue life and damage results.

For this workflow, when you create an NX Nastran SEMODES 103 — Flexible Body solution for flexible multibody durability analysis, you must select **Export** from the **Flexible Body Solution Type** list on the **Case Control** tab in the **Solution** dialog box.

In the **Transient Durability Event** dialog box, when you select an NX Nastran SEMODES 103 — Flexible Body solution with export option from the

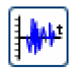
Transient Solution List, you must also specify the corresponding MDF file in the **MDF File** box.

In previous releases, you could only perform durability analysis on stress or strain results from an NX Nastran SEMODES 103 — Flexible Body solution with recovery option. Its OP2 file contains stress and strain histories that the durability solver can directly use.

Why should I use it?

This workflow allows you to analyze durability results from large models as the OP2 file contains only modal results and can hence be considerably smaller in size.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Results from an NX Nastran SEMODES 103 — Flexible Body solution with export option (OP2 file) and results from the corresponding flexible body motion solution (MDF file)
Toolbar	**Durability® Transient Durability Event** 
Menu	**Insert® Durability® Event® Transient**
Simulation Navigator	Right-click the Durability solution process node® **New Event® Transient**

Static offset for transient events

What is it?

You can specify a static structural solution as the static offset for the transient durability event.

At each element, the static stress or strain is added to the transient stress or strain history.

Why should I use it?

By specifying thermal solution or static solution as the static offset, you can superpose static loads upon the transient response of the structure when you compute life and damage results to model thermal or structural pre-loads.

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	**Static Offset** table list

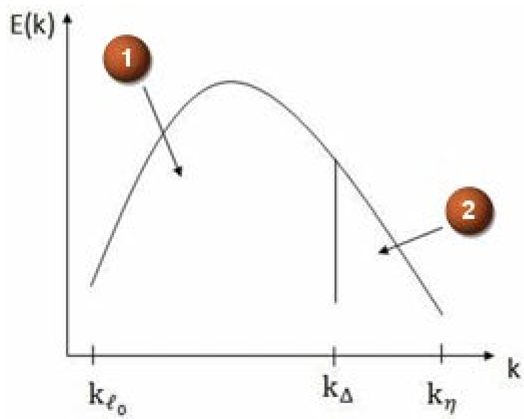
NX Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal

Large Eddy Simulation (LES)

What is it?

Use the new **LES - Large Eddy Simulation** turbulence model to resolve large eddies when you simulate turbulent fluid flow. LES solves the filtered Navier-Stokes equations. All other available turbulence models in flow solvers model turbulent structures using Reynolds stress terms in the Reynolds-Averaged Navier-Stokes (RANS) equations.

Instead of trying to resolve the entire energy spectrum of turbulence, LES defines a grid that can capture the largest turbulent scales (eddies) in the flow down to a spatial scale Δ, called the cutoff length scale. The theory states that these large eddies are dependent on the nature of the flow and its boundary conditions. They represent the left-hand side of the turbulent energy spectrum (1). On the other hand, the smallest eddies (smaller than the cutoff length scale) can be modeled by taking advantage of the fact that they are homogenous and universal in any turbulent flows. These small eddies (2) are modeled by introducing a physical or empirical model called a subgrid scale model that can account for their effect in the flow.



Turbulent energy spectrum — large scales are resolved (1), small scales are modeled (2)

The resulting velocity field is a filtered velocity field where the small, subgrid, scales are approximated using one of the following models:

Smagorinsky-Lilly model	Assumes that the energy production and dissipation of small scales are in equilibrium. This is the oldest and most used LES subgrid scale model.
WALE model	Takes into account the dissipative effect of the turbulent structures with a high rate of deformation, a high rate of rotation, or both and reproduces proper asymptotic variation of turbulent viscosity close to the wall. WALE stands for Wall Adapting Local Eddy-viscosity.
Vreman model	Properly captures the transition of flows from a laminar to a turbulent regime.

When you use the LES turbulence method, it is recommended that you:

- Select one of the high-order advection schemes on the **3D Flow Solver** tab of the **Solver Parameters** dialog box.
- Use the **Convective Outflow** type of the **Flow Boundary Condition** simulation object to specify the outlet boundary conditions on your fluid domain.
- Select **Fractional Step** from the **Parallel Flow Solver Scheme** list.
- Use a very small timestep to resolve the turbulence.

Why should I use it?

The **LES - Large Eddy Simulation** turbulence model is recommended when you want to visualize the actual eddies in the flow and when you are interested in the instantaneous flow fields and their statistics.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow Coupled Thermal-Flow	Advanced Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation

Prerequisite	You must set the **Solver Selection** to **Parallel Solver**.
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® **Turbulence Model** list® **LES - Large Eddy Simulation**

Convective Outflow flow boundary condition

What is it?

REVIEW NOTE

Issue: This topic is under construction

Use the new **Convective Outflow** type of **Flow Boundary Condition** simulation object to model fluid as an outlet boundary condition without specifying the pressure.

Note You cannot specify a **Convective Outflow** in a model that already has an **Opening** type of **Flow Boundary Condition** simulation object.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow Coupled Thermal-Flow	Advanced Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	On the **Solution Details** tab in the **Solution** dialog box, you must set: • **Solver Selection** to **Parallel Solver** • **Solution Type** to **Transient**
Toolbar	**Advanced Simulation** ® **Flow Boundary Condition** 
Simulation Navigator	Right-click the **Simulation Object** container node ® **New Simulation Object** ® **Flow Boundary Condition**
Location in dialog box	**Type** list ® **Convective Outflow**

Non-geometric elements

What is it?

Use the new **Non-Geometric Element** (NGE) modeling object to fix either a temperature or a capacitance that is not associated with a physical location. If you fix the capacitance, you can model phase change within the NGE.

A non-geometric element is a calculation point in thermal models. It is not associated to a node or geometry and cannot be displayed graphically.

The following table lists modeling objects, simulation objects, loads, and constraints in which you can use **Non-Geometric Element** modeling objects.

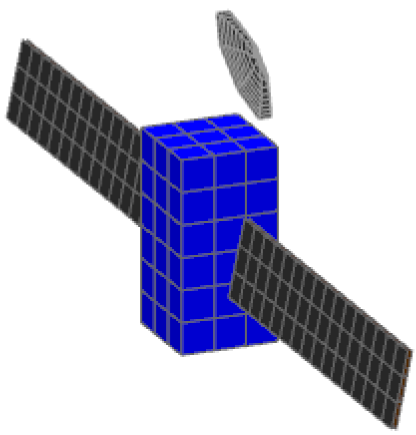
Name	Type
Modeling objects	
Active Heater Controller	All
Generic Entity	All
Thermostat	All
Simulation objects	
Thermal Coupling	**Total Conductance**, only when the **Distribution** list is set to **Per Element**. **Total Resistance** **Heat Transfer Coefficient** **Edge Contact**
Thermal Coupling – Advanced	**User Function**

Report	**Per Element** **Between Regions** **Heat Maps**
Loads	
Thermal Load	**Heat Load**
Constraints	
Initial Conditions	**Initial Temperature**

Why should I use it?

You can use non-geometric elements to model an object at a fixed temperature or fixed capacitance without having to create the geometry and the elements.

For example, you could use a non-geometric element at a fixed temperature to model the contents inside a spacecraft. You could then couple the non-geometric element to the six surfaces of the main unit with a thermal coupling. By using a non-geometric element and a thermal coupling, you model the thermal behavior without creating several elements to represent the contents in the main unit of the spacecraft.



Spacecraft meshed with 2D elements

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

NX Thermal and Flow	Thermal	Thermal Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

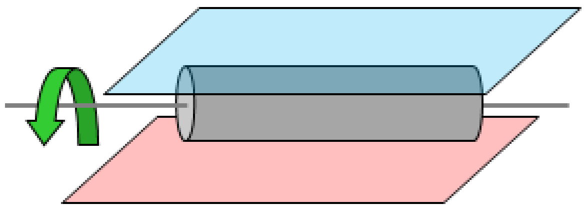
Application	Advanced Simulation
Toolbar	**Advanced Simulation® Modeling Objects**
Menu	**Insert® Modeling Objects**
Location in dialog box	**Type** list® **Non-Geometric Element® Create**

Spinning

What is it?

Use the new **Spinning** type of **Solid Motion Effects** simulation object to model the thermal effects of a model with a spinning component.

A spinning component spins around a specified axis much faster than the thermal time constants of the model. Because there is no time for the model to react thermally to one full spin rotation, the effects of the spinning are averaged over the rotation.



Cylinder spinning between a hot and a cold plate

The thermal solver calculates and averages around the spin axis all view factors and thermal couplings that are defined in the model. The view factors include:

- Solar, earth, and albedo view factors from orbital heating
- Heat flux view factors from radiative heating
- Black body and ray-traced view factors from the deterministic method
- Ray-traced view factors, radiative couplings, and heat loads from the Monte Carlo method

Note **Spinning** does not work with the hemicube method for Beta, but is planned to work for the final release.

In previous releases, you could only define spinning for orbits, and the thermal solver averaged only solar, earth, and albedo view factors at each spin position.

When you use the **Spinning** type of **Solid Motion Effects** simulation object, you must select the region that is spinning, the **Spinning Axis**, and specify the **Number of calculations per spin**, N. The thermal solver calculates the view factors and thermal couplings due to spinning N times, each time rotating the spinning region by 360/N degrees. The thermal solver then averages the view factors and thermal couplings of these N positions.

Note For thermal coupling calculations only, best accuracy is obtained when N is greater than the number of elements around the circumference of the spinning part.

Why should I use it?

Spinning lets you model heat transfer of a spinning object, like a rotisserie chicken in a closed barbecue grill.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal Coupled Thermal-Flow	Advanced Thermal Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Solid Motion Effects**
Simulation Navigator	Right-click the **Simulation Object** container node ® **New Simulation Object** ® **Solid Motion Effects**
Location in dialog box	**Type** list ® **Spinning**

Solid Motion Effects

What is it?

The **Articulation** simulation object found in the previous releases is now located in the **Solid Motion Effects** simulation object as the **Articulation** type.

With the **Solid Motion Effects** simulation object, you can also define the **Spinning** type.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** ® **Solid Motion Effects**
Simulation Navigator	Right-click the **Simulation Object** container node ® **New Simulation Object** ® **Solid Motion Effects**

Fully coupled pressure-velocity and fractional step schemes in the parallel flow solver

What is it?

You can now choose between the following two flow solver schemes for the parallel flow solver:

Fully Coupled Pressure-Velocity Solves together mass and momentum equations by iterating until convergence is achieved in each timestep or steady state iteration.

Fractional Step Solves mass and momentum equations separately once per timestep or steady state iteration.

In the previous release, the fractional step scheme was only used by the parallel flow solver and the fully coupled pressure-velocity scheme was only used by the serial flow solver. The serial flow solver still only uses the fully coupled pressure-velocity scheme.

Why should I use it?

The **Fully Coupled Pressure-Velocity** scheme is better suited for steady-state models or transient models with large timestep. The **Fractional Step** scheme is suitable for transient models with a small timestep. The fractional step scheme is faster per timestep or iteration and uses less memory than the fully coupled pressure-velocity scheme.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow Coupled Thermal-Flow	Advanced Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must set the **Solver Selection** to **Parallel Solver**.
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® **Parallel Flow Solver Scheme** list

Semi-implicit Second-order time integration method

What is it?

You can use the new **Semi-implicit Second-order** time integration method to discretize flow field equations in the time domain when you select the parallel flow solver.

You set this option in the new **Flow Time Integration Method** list that also lets you select the **Implicit** time integration method, the default method. In previous releases, the flow solver used the implicit time integration method.

Because the **Semi-implicit Second-order** method is a second-order accurate temporal discretization, it allows the accurate resolution of transient flow features with a significantly larger timestep, than the **Implicit** method that is a first-order accurate temporal discretization.

Why should I use it?

Use the **Semi-implicit Second-order** method when you model flow fields using the LES turbulence model as LES simulations must accurately capture flow structures with a wide range of associated time scales.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow Coupled Thermal-Flow	Advanced Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must set the **Solver Selection** to **Parallel Solver** and the **Solution Type** to **Transient** on the **Solution Details** tab.
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	**Transient Setup** tab® **Time Integration Control** group® **Flow Time Integration Method** list

Second-order (CDS) advection scheme

What is it?

A new second-order advection scheme for the flow solvers is now available. Use the **Second-order (CDS)** advection scheme to specify the central differencing scheme (CDS) as the numerical discretization method to solve the partial differential equations that describe the flow.

CDS provides a more accurate representation of the advective fluxes on a given mesh than does the first-order upwinding differencing scheme (UDS). CDS eliminates the severe smearing of the solution field arising with UDS on meshes which are not well-aligned with the local flow direction.

Why should I use it?

The CDS scheme is computationally less expensive than the existing QUICK and SOU second-order schemes. CDS is a central scheme which is optimal for low speed flows with Peclet number less than 10.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC

NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
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® **Advection Schemes** group® **Momentum** or **Energy** or **Two-Equation Turbulence Model** or **Humidity, Tracer Fluids and Homogeneous Mixtures** lists® **Second-order (CDS)**

Automatic limiters for second-order advection schemes

What is it?

When you specify a second-order advection scheme, you can now select between two methods that the flow solver uses to automatically compute the limiter value. The flow solver uses the limiter to compute the advective flux as a weighted average of the first-order scheme approximation and second-order scheme approximation.

Automatic limiter methods compute the limiter value by searching for extreme values over a region surrounding a given node. The two automatic limiter methods are:

General Convective Boundedness Condition — This method searches for the extreme values over all neighboring nodes of a given node.

Stabilized Convective Boundedness Condition — This method searches for the extreme values over neighboring nodes that are upwind of the node of interest.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type

NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must set at least one of **Momentum** or **Energy** or **Two-Equation Turbulence Model** or **Humidity, Tracer Fluids and Homogeneous Mixtures** lists to **Second-order (QUICK)** or **Second-order (SOU)** or **Second-order (CDS)** and **Limiter** list to **Automatic**.
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® **Advection Schemes** group® **Momentum Limiter Stabilization** or **Energy Limiter Stabilization** or **Two-Equation Limiter Stabilization** or **Humidity, Tracer Fluids and Homogeneous Mixtures Limiter Stabilization**

Static Pressure Flow Boundary Condition

What is it?

Use the new **Static Pressure** type of **Flow Boundary Condition** simulation object to model the fluid with a known static pressure at a location.

The **Static Pressure** flow boundary condition acts either as an inlet or an outlet depending on the physics of the model. For example, to model pulsatile flows, such as the blood flow in an artery, you can specify the known static pressure as a function of time at one end of the artery.

Note The specified static pressure value must be an absolute pressure.

Why should I use it?

You can use the **Static Pressure** flow boundary condition for pressure-driven flows with specified static pressure.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow Coupled Thermal-Flow	Advanced Flow Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must select **Parallel Solver** from the **Solver Selection** list on the **Solution Details** tab in the **Solution** dialog box.
Toolbar	**Advanced Simulation** ® **Flow Boundary Condition** 
Simulation Navigator	Right-click the **Simulation Object** container node ® **New Simulation Object** ® **Flow Boundary Condition**
Location in dialog box	**Type** list ® **Static Pressure**

Parametric optimization in NX Thermal and Flow

What is it?

REVIEW NOTE

Issue: This topic is under construction

Enable Multithreading option

What is it?

Use the new **Enable Multithreading** option to allow shared-memory parallelization in the Analyzer module of the thermal solver.

The Analyzer module is the module in the thermal solver which calculates steady state or transient temperatures and total pressures for models with thermal and duct elements.

When you enable multithreading, you must specify the number of execution threads. To optimize the use of your computer, it is recommended that you set the number of execution threads to the number of cores or processors on your computer. For example, set the number of execution threads to two for dual-core machines or to four for quad-core machines.

Why should I use it?

Multithreading takes advantage of multi-core and multi-processor hardware and speeds up the thermal computations. Multithreading is most-useful for radiation-dominated models.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
	Axisymmetric Thermal	Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	(**Coupled Thermal-Flow** analysis types) You must select the **Solve Thermal** check box on the **Solution Details** tab in the **Solution** dialog box.
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® **Parallel Processing** group® **Enable Multithreading** check box

Teamcenter Integration for Simulation

Results JT creation and export

What is it?

Note This topic is under construction.

In previous releases, and JT files created from solver results displayed in a post view would have to manually imported into a DirectModel dataset in the Teamcenter client. Beginning with this release, you can generate JT files of results visualizations and automatically import them into Teamcenter directly from Advanced Simulation.

To generate results JT files, select one or more results components in the **Post Processing Navigator**, right-click, and choose **New JT File**. In the **JT Post Display Export** dialog box, you can click to rename the JT file and dataset.

By default, NX exports each JT file to a unique dataset.
You can change the default behavior by selecting the **Simulation→General→Teamcenter→Aggregate Post JTs into One Dataset** customer default.

Where do I find it?

Application	Advanced Simulation
Prerequisite	
Toolbar	
Menu	®

CAE usability enhancements

What is it?

Note Several enhancements have been added to improve general usability when working with CAE data in Teamcenter Integration:

- **Simulation File View** enhancements: The Teamcenter name, description and status are now displayed in the **Simulation File View** for all loaded revisions. You can right-click any Simulation (CAEAnalysis revision) in the Simulation File and choose Show Results Files to view all related solver results files.

- **Browse** enhancements: When you access the Teamcenter browser through a dialog box, the items displayed are filtered to show only the appropriate CAE item types. For example, when you use the **Append FEM** command, only CAEModel items are shown. When you use the **Import Results** command, only CAEAnalysis items are shown.

- **Open Associated Model**: When creating an assembly FEM, on the Map CAD Component to Existing FEM dialog box, click **Open Associated Model** [icon] to choose from a list of all CAEModel revisions associated with the CAD item revision.

- The standard NX **Create Clone Assembly** command is now available in Advanced Simulation. This is especially useful when revising or copying an assembly FEM.

Where do I find it?

Create Clone Assembly

Application	Advanced Simulation
Menu	**Assemblies® Cloning® Create Clone Assembly**

Open Associated Model

Application	Advanced Simulation
Prerequisite	An associative assembly FEM with ignored CAD components.
Simulation Navigator	Right-click ignored CAD component® **Map Existing® Open Associated Model** [icon]

Application	Advanced Simulation
Prerequisite	.

Simulation Navigator	
Menu	® ®

NX 8.0.1 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:
  - 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.
  - 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 8.0.1
NX Nastran	Import ASCII (.dat)	8	8
	Import Binary (.op2)	8	8
	Export ASCII (.dat)	8	8
	Post-processing of Results (.op2)	8	8.1

Solver	File Type	NX 8	NX 8.0.1
MSC Nastran	Import ASCII (.dat)	2011.1	2011.1
	Import Binary (.op2)	2011.1	2011.1
	Export ASCII (.dat)	2011.1	2011.1
	Post-processing of Results (.op2)	2011.1	2011.1
Abaqus	Import ASCII (.inp)	6.10	6.10
	Import Binary	N/A	N/A
	Export ASCII (.inp)	6.10	6.10
	Post-processing of Results (.fil)	6.11	6.11
	Post-processing of Results (.odb)	6.10-EF1	6.11
ANSYS	Import ASCII (PREP7, CDB)	13	13
	Import Binary (.rst, .rth)	13	13
	Export ASCII (.inp)	13	13
	Post-processing of Results	13	13
LS-DYNA	Import ASCII	971R5.0	971R5.0
	Import Binary	N/A	N/A
	Export ASCII (.k)	971R5.0	971R5.0
	Post-processing of Results	971R5.0	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 7.5.4	NX 7.5.5.
NX Nastran	Import ASCII (.dat)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Import Binary (.op2)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Export ASCII (.dat)	6.1	7.0	7.0	7.1	7.1	7.1	8
	Post-processing of Results	6.1	7.0	7.1	7.1	7.1	7.1	8

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2	NX 7.5.3	NX 7.5.4	NX 7.5.5.
MSC Nastran	Import ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Import Binary (.op2)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Export ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
	Post-processing of Results	2008r1	2008r1	2008r1	2008r1	2010	2010	2011.1
Abaqus	Import ASCII (.inp)	6.8-1	6.9–1	6.9–1	6.9-1	6.10	6.10	6.10
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9	6.9	6.9	6.10	6.10	6.10
	Post-processing of Results (.fil)	6.8-EF2	6.9.2	6.9.2	6.10-1	6.10-1	6.10-1	6.11-1
	Post-processing of Results (.odb)	6.8-EF2	6.9-EF1	6.9-EF2	6.9-EF2	6.10-EF1	6.10-EF1	6.10-EF1
ANSYS	Import ASCII (PREP7, CDB)	12	12.1	12.1	12.1	13	13	13
	Import Binary (.rst, .rth)	12	12.1	12.1	12.1	13	13	13
	Export ASCII (.inp)	12	12.1	12.1	12.1	13	13	13
	Post-processing of Results	12	12.1	12.1	12.1	12.1	12.1	12.1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1

NX 6 releases

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3	NX 6.0.4	NX 6.0.5
NX Nastran	Import ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Import Binary (.op2)	6.0	6.1	6.1	6.1	6.1	7.0
	Export ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Post-processing of Results	6.0	6.0	6.1	6.1	7.0	7.0
MSC Nastran	Import ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Import Binary (.op2)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Post-processing of Results	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
Abaqus	Import ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Post-processing of Results (.fil)	6.7-5	6.8-1	6.8-3	6.8-EF2	6.8-EF2	6.8-EF2
	Post-processing of Results (.odb)	N/A	N/A	N/A	6.8-EF	6.8-EF2	6.9-EF2
ANSYS	Import ASCII (PREP7, CDB)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Import Binary (.rst, .rth)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Export ASCII (.inp)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Post-processing of Results	11 SP1	11 SP1	11 SP1	11 SP1	12.0	12.1

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
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

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
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

Mesh Display usability improvements

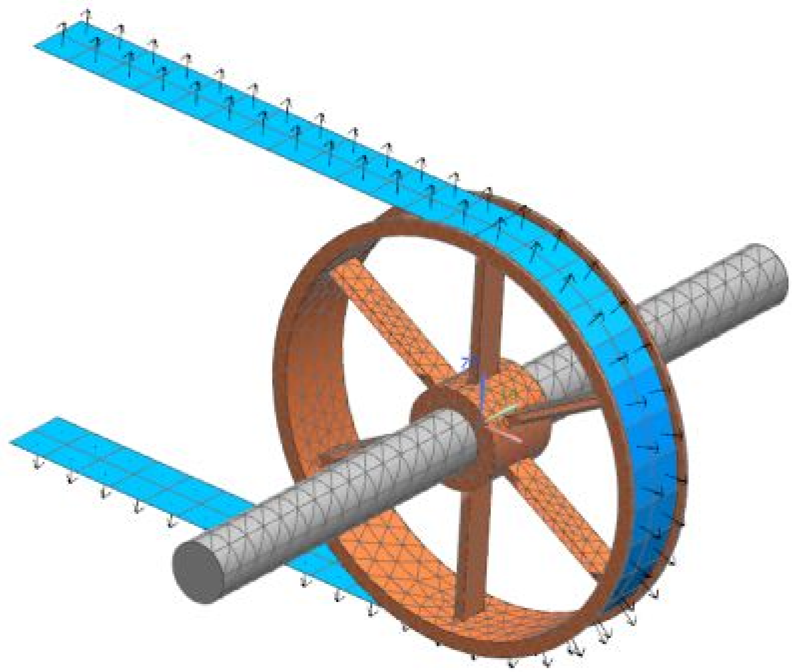
What is it?

With this release, it is now easier to edit the display attributes of meshes and mesh collectors. The **Mesh Display** dialog box has been redesigned. Display attributes for element types are separated into tabs by element family (3D, 2D, 1D, and 0D). You can now set display attributes for selected meshes, all meshes and collectors, or all meshes in a selected collector.

Display attributes include the following:

- Element face and edge color
- Solid element internal edges
- Element normals
- 2D thicknesses
- Beam cross sections, orientation, and end releases

Default mesh display attributes are now defined only in the **Customer Defaults** dialog box.



Where do I find it?

Customer default

Application	Advanced Simulation, Design Simulation
Menu	File® Utilities® Customer Defaults
Location in dialog box	**Customer Defaults** dialog box® **Simulation**® **Mesh Display**

Command

Application	Advanced Simulation, Design Simulation
Menu	**Preferences**® **Mesh Display**
Simulation Navigator	Right-click a mesh collector, one mesh, or multiple meshes® **Edit Mesh Display**

New non-manifold element edge display

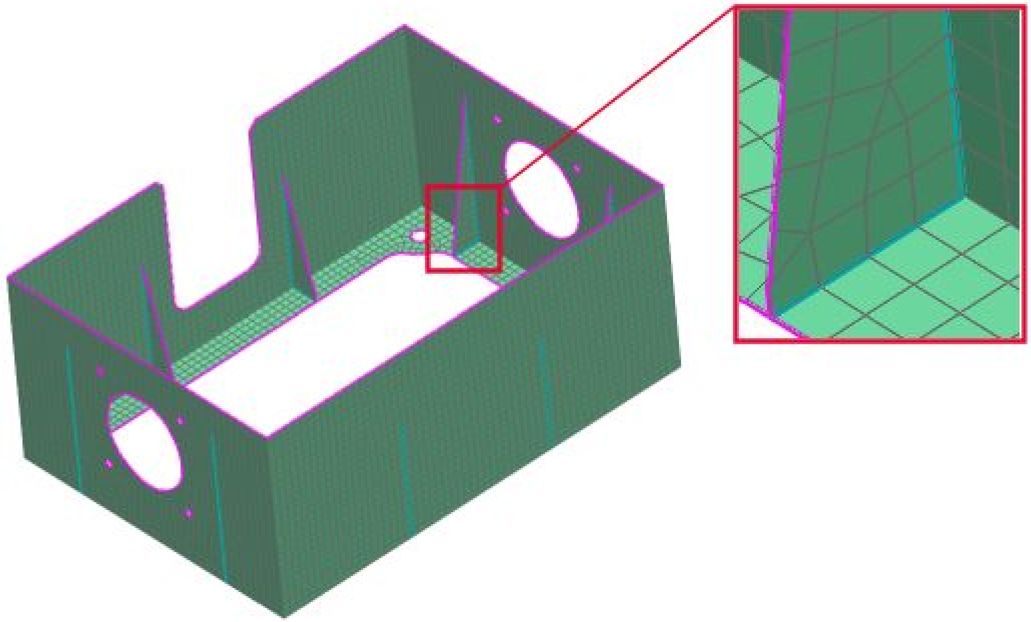
What is it?

You can now use the **Element Outlines** option in the **Model Check** dialog box to highlight any non-manifold element edges in your model as well as any free element edges.

When you select the **Display Free and Non-Manifold Edges** option, the software evaluates 2D elements and 3D element free faces for free and non-manifold edges. The software creates a temporary display and highlights:

- Any elements with free edges in magenta.

- Any non-manifold element edges in cyan.



Understanding element free edges

A free element edge is an edge that is referenced by only one element. Free element edges in the interior of a sheet body, for example, can indicate regions of your model that are not fully connected (stitched).

Understanding non-manifold element edges

A non-manifold element edge is an edge that is shared by more than two elements. Non-manifold element edges are created when the underlying geometry is non-manifold. For example, in Advanced Simulation, when you generate a midsurface on a part that contains ribs, non-manifold geometry conditions can be created . You can then use the **Stitch Edge** command to ensure that the non-manifold faces are fully stitched together prior to meshing.

The ability to display all non-manifold element edges is helpful when you need to validate that the faces in your model are fully stitched before you solve. When you examine the areas in your model where non-manifold faces intersect, you can use the **Display Free and Non-Manifold Edges** option to verify that the elements on those intersecting faces are also stitched together.

Note With 2D elements, when you use the **Display Free and Non-Manifold Edges** option, you should first clear the **Element Thickness and Offset** check box in the **Mesh Collector Display** or **Mesh Display** dialog box. The thickness and offset display can make it hard to visualize the temporary graphics used by the **Display Free and Non-Manifold Edges** option.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with one or more existing 2D meshes
Toolbar	**Advanced Simulation® Model Check**
Menu	**Analysis® Finite Element Model Check**

New check for duplicate elements

What is it?

You can use the new **Duplicate Elements** option in the **Model Check** dialog box to check your model for elements that share the same nodes.

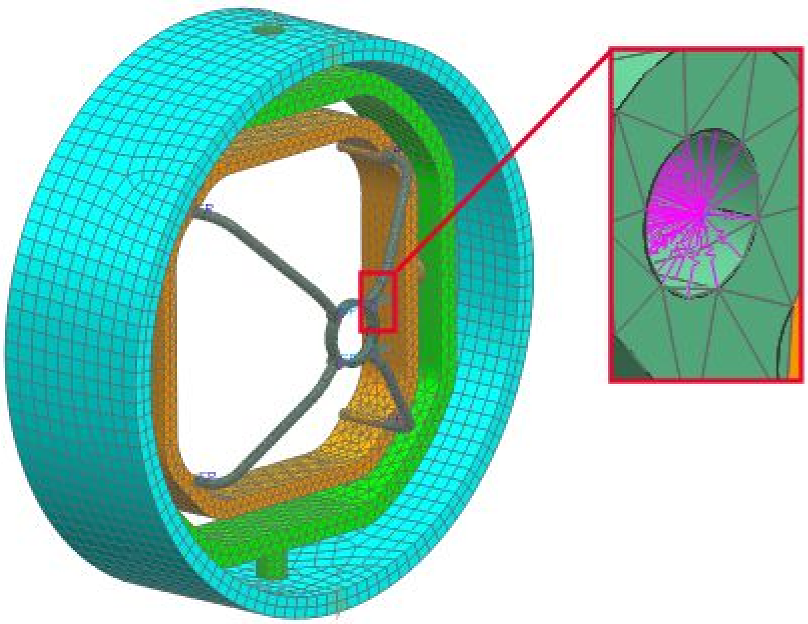
With the **Duplicate Elements** option, the software evaluates all elements that have the same topology (for example, 1D, 2D, 3D). This means, for example, that the software considers a Nastran CBEAM element and a 2–noded RBE2 element to be duplicates when they exist at the same location and used the same pair of nodes to define their connectivity.

If the software detects any duplicate elements, the software:

- Temporarily highlights the duplicate elements in magenta in the graphics window.
- Lists the IDs of all duplicate elements and the shared (vertex) nodes in the **Information** window.
- Places all duplicate elements into an output group in the **Simulation Navigator**.

The **Duplicate Elements** option is helpful when, for example, your model contains multiple 1D connections. You can use the **Duplicate Elements** option to validate that those 1D connections are defined appropriately before you solve.

For example, in the Nastran environment, a CELAS1 element is defined with two nodes, each with one DOF. If you connect two different components that are meshed with solid elements with CELAS1 elements, you need three CELAS1 elements defined on the same set of nodes to fully define the connection. In the following graphic, the **Duplicate Elements** option was used to identify multiple, coincident CELAS1 elements in a larger assembly FEM.



Deleting duplicate elements

If the software identifies any duplicate elements, you can optionally choose to delete one of those elements. In the **Model Check** dialog box, you can use the **Element Keep Preference** list to specify whether you want the software to retain the element with the higher or lower label (ID).

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with one or more existing meshes
Toolbar	**Advanced Simulation® Finite Element Model Check** 
Menu	**Analysis® Finite Element Model Check**

Improved capabilities for detecting interference and clearance issues

What is it?

This release includes enhancements that make it easier for you to detect interference and clearance issues in your model. These enhancements include:

- A new option in the **Model Check** command that allows you to check all visible faces in your FEM file for interference or clearance issues before you solve.

- An automatic interference check in the **3D Tetrahedral Mesh** command that automatically checks fluid bodies for interfering faces.

New model check option for interference and clearance issues

You can use the new **Detect Interference/Clearance** option in the **Model Check** dialog box to evaluate your model for regions where:

- Faces intersect each other.
- The clearance between faces is too small relative to the element size that you want to use for solid meshing.

The software checks for either interference or clearance between faces depending the **Clearance** value you specify.

- If you specify a **Clearance** value of 0.0, the software looks for intersecting faces.
- If you specify a **Clearance** value > 0.0., the software looks for faces where the distance between two faces is less than this specified value.

With the **Detect Interference/Clearance** option, the software evaluates all visible bodies in your model for interference or clearance issues. If the software finds any intersecting faces or faces that lie within the specified clearance tolerance, the software:

- Places those faces in an output group in the **Simulation Navigator**.
- Creates a temporary display that shows the facets on those faces.

Checking for intersecting faces

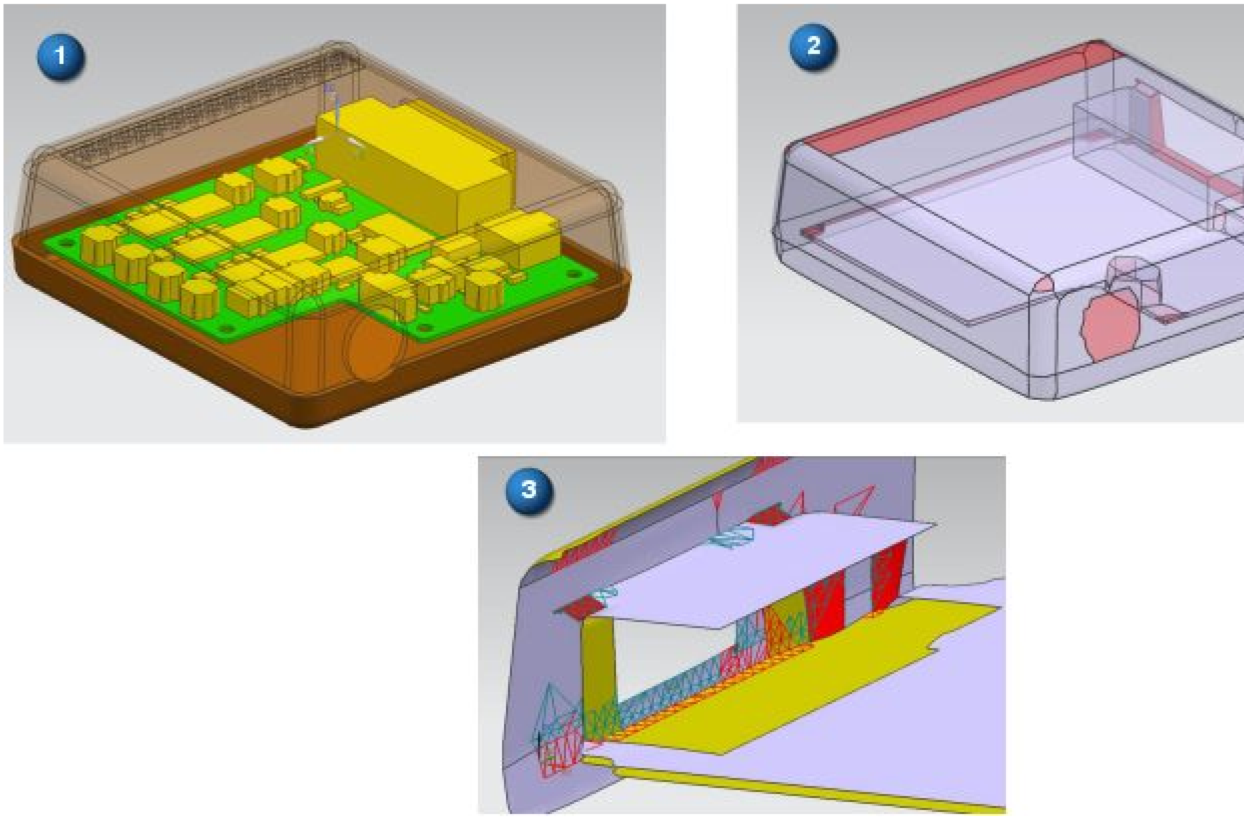
If you specify a **Clearance** value of 0.0, the software looks for intersecting faces between all visible:

- Solid bodies.
- Sheet bodies.
- Solid and sheet bodies.

For example, your model may contain intersecting faces if:

- Assembly components are poorly positioned.
- You used the **Surface Wrap Fluid Domain** command to create a fluid body and did not specify a sufficiently precise value in the **Global Resolution** box. If the **Global Resolution** value you specify is too large, the tessellation on the resulting fluid body may be too coarse.

  The following graphic shows an example of intersecting faces on a fluid body. (1) shows the original solid body. (2) shows the fluid body created by the **Surface Warp Fluid Domain** command. (3) shows the intersecting faces that the **Detect Interference/Clearance** option found on the fluid body.

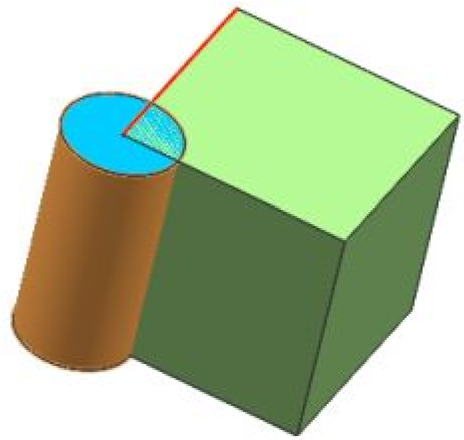


Using the **Detect Interference/Clearance** option to check for interference issues can help you locate regions where the interferences are significant enough that you must resolve the issue on the original CAD geometry in the Modeling task.

- Where the interference between two intersecting sheet bodies is significant enough that you cannot use the **Stitch Edge** command to eliminate the overlap.

- Where the interference between two intersecting solid bodies is significant enough that it cannot be resolved with the **Mesh Mating Conditions** command.

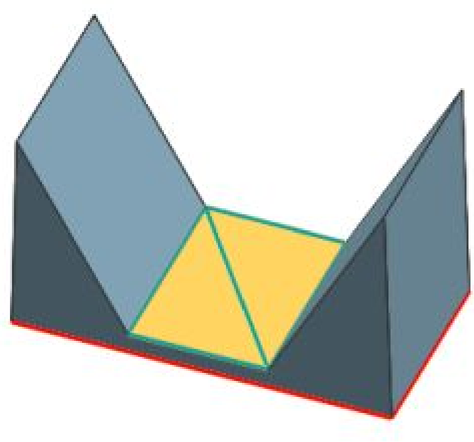
The following graphic shows how the software highlights the intersecting faces on two solid bodies.



Checking for clearance issues

You can also use the **Detect Interference/Clearance** option to ensure that there is adequate clearance in your model for the size of element you plan to use in the mesh. For example, in a model that contains very thin or tight regions, you could use the **Detect Interference/Clearance** option with a clearance value set to the solid element size you want to use to evaluate the appropriateness of that element size.

In the following example, we used the **Detect Interference/Clearance** option with a clearance value of 15. Notice how the software detected a clearance issue with the highlighted faces.



Automatic interference checking for fluid bodies during tetrahedral meshing

Now, when you use the **3D Tetrahedral Mesh** command to generate a mesh on a fluid body, the software automatically checks the body for any intersecting faces before it generates any elements. If the software detects any intersecting faces within the fluid body, it does not try to generate a mesh on the body. However, it proceeds with meshing any additional selected bodies.

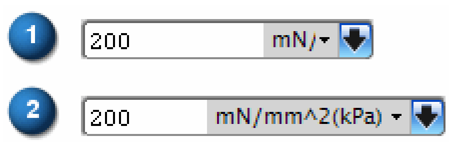
Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® Finite Element Model Check**
Menu	**Analysis® Finite Element Model Check**

Full display of units label

What is it?

Beginning with this release, in Advanced Simulation and Design Simulation, the units label in most dialog boxes has been expanded to accommodate the entire unit description. This enhancement has not been applied to meshing dialog boxes.



(1) Pre-NX8.0.1 units label; (2) New units label

Support added for the SC03 solver

What is it?

This release includes initial support for the proprietary SC03 solver. You can use the new SC03 solver to:

- Create and edit FEM and assembly FEM files.
- Automatically generate meshes.
- Export SC03 input (.pm) files.

NX does not currently support:

- Loads and boundary conditions for models in the SC03 environment.
- The import of SC03 files.

Geometry idealization and abstraction

Support for the Redo command in the idealized part

What is it?

The **Undo** command is now available when you are working with an idealized part in **Advanced Simulation**. You can use the **Redo** command to reapply the operation that you reversed with the **Undo** command.

- **Redo** is only available immediately after you use the **Undo** command.
- **Redo** is not supported for all commands. Redo is supported for all commands that can be journaled.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active idealized part file
Toolbar	**Standard® Redo**
Menu	**Edit® Undo List/Redo**

Midsurface enhancements

What is it?

This release includes the following improvements to the **Midsurface by Face Pairs** command:

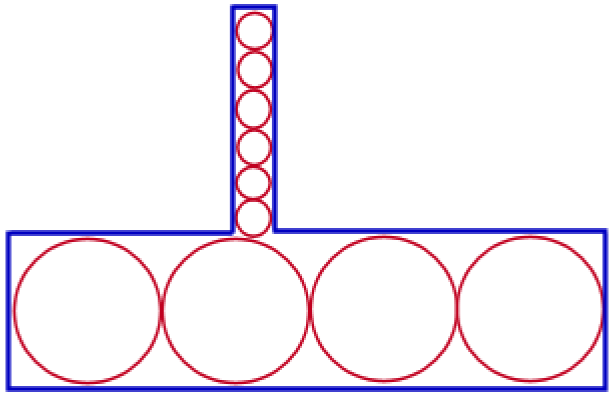
- Improved thickness calculation algorithm for **Progressive** pairing.
- Automatic pairing now available for tangent-continuous parts.
- Ability to create face pairs based on the ratio of thickness to face size.
- Additional improvements for special cases.

Note Because of the extent of these enhancements, if you created an NX Open program in NX 7.5 in which you use the **Midsurface by Face Pairs** command, you will need to recompile that program in NX 8.0.1.

Improved thickness calculation algorithm for Progressive pairing

When you use the **Progressive** pairing strategy, the software now uses a rolling ball thickness calculation algorithm to create face pairs.

A rolling ball calculation is analogous to using a moving sphere, similar to an inflatable ball, that is constrained within the walls of the solid body. The ball contacts a solid face at one point and expands until it contacts the adjacent faces, creating the largest ball that can fit within the constraints of the faces. The diameter of the ball becomes larger or smaller automatically as it rolls through the different regions of the part. The software then uses the diameter of the ball at a given location as the thickness of the part at that location.



In previous releases, the **Progressive** strategy created face pairs based on a single, computed average thickness value for each body. While this technique was useful for bodies where the thickness was fairly constant, it was not accurate for variable thickness parts.

Because the rolling ball method provides more granular thickness information throughout a body, it provides more accurate thickness data for both constant and variable thickness parts. This results in more accurate face pairs.

Automatic pairing for now available for tangent-continuous parts

Both the **Progressive** and **Thickness** pairing strategies have been improved to handle the creation of midsurfaces on tangent-continuous parts. In a tangent continuous part, the angle between faces is less than 3°. In previous releases, the best practice for creating a midsurface on a tangent continuous part was to use the **Manual** pairing strategy.

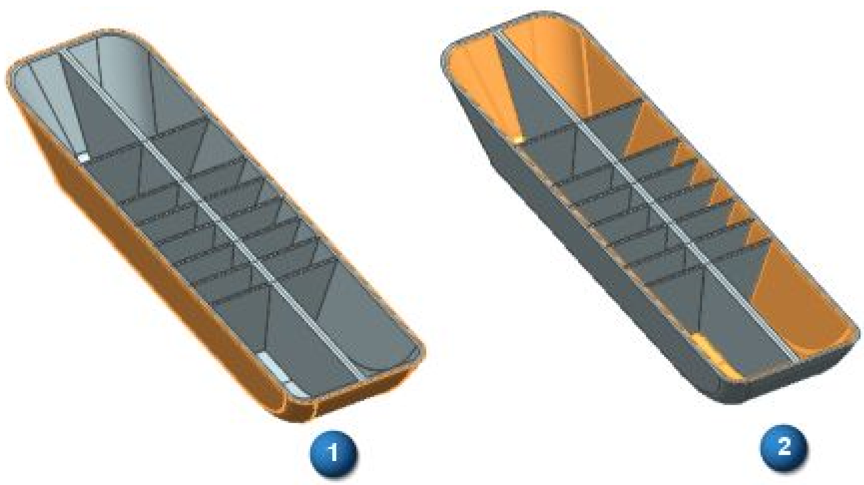
You can use the new **Merge Angle Tolerance** option to merge face pairs. Merging face pairs:

- Significantly reduces the overall number of face pairs that you have to manage.
- Decreases the number of individual mid-sheets that you must stitch together.

The **Merge Angle Tolerance** option merges face pairs when the angle between the individual pairs is less than the value you specify.

For example, consider the tangent-continuous model shown below. In the NX 8 release with the **Progressive** pairing strategy, the software generated a total of 36 face pairs for this model, including 13 individual face pairs for the main outer and inner surfaces. Some of those 13 individual face pairs are show below.

In this release, with the **Progressive** pairing strategy and a **Merge Angle Tolerance** of 5°, the software generates a total of 9 face pairs, with one single pair for the main outer (1) and inner (2) surfaces, as shown below.



Ability to create face pairs based on the ratio of thickness to face size

With either the **Progressive** or **Thickness** pairing strategies, you can use the new **Thickness Ratio (D/T)** to have the software create face pairs based upon the sizes of faces relative to their thickness. With this option, the software divides the smallest characteristic length of a face (D) by the maximum local thickness (T) between the two faces to be paired.

- If you decrease the **Thickness Ratio (D/T)** value, the software finds more face pairs.

- If you increase the **Thickness Ratio (D/T)** value, the software finds fewer face pairs.

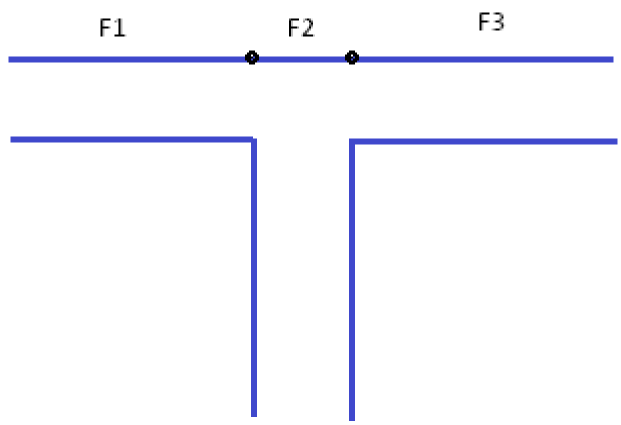
The **Thickness Ratio (D/T)** option is particularly helpful for models with highly variable thickness values. It is also helpful for models where you want to ignore (not pair) small faces, such as the faces that comprise small fillets.

Additional improvements for special cases

This release also includes the following general improvements to the **Midsurface by Face Pairs** command.

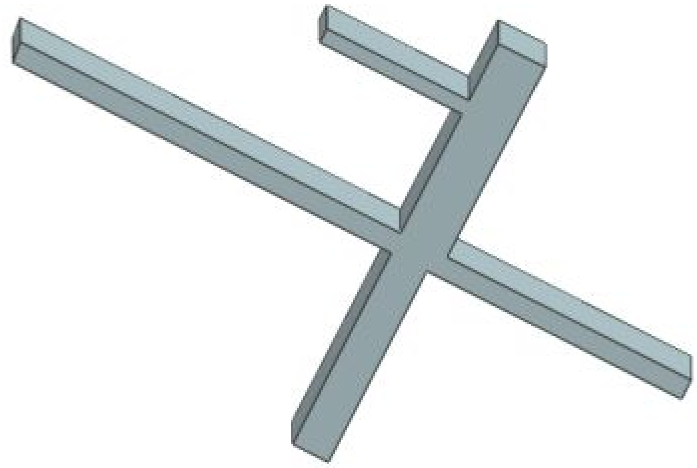
- The software now adds planar faces that are missing a pairing face to side 1 of an adjacent face pair. This occurs when the size of the face that has no pair is less than the average thickness value.

  In the example below, F2 does not have an opposing face. In previous releases, NX would not have included F2 as a part of any face pair. Now, because the size of F2 is smaller than the average thickness in that region, the software merges F2 into side 1 of a face pair with F1 and F3.

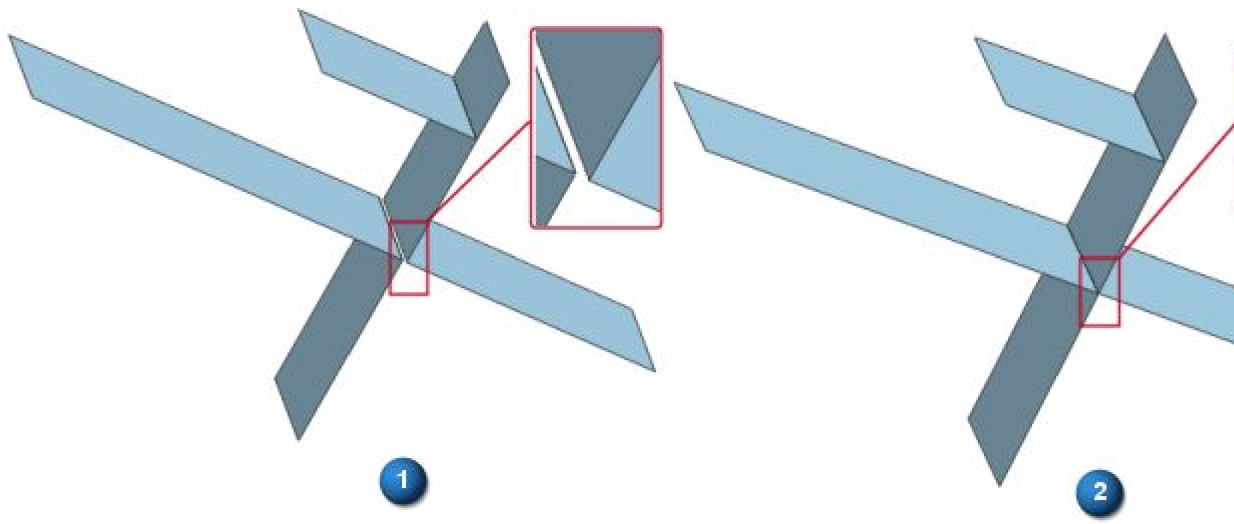


- The software now generates a *floating* midsurface solution for face pairs that are slightly offset from each other due to variations in the thickness of the part. This allows the software to create connected mid-sheets even when the theoretical mid-sheets for individual face pairs do not align perfectly.

  For example, consider the part shown below, which is comprised of a series of intersecting bars. Note that the bars vary in their thickness.



The graphic below shows the differences in the midsurfaces that the software creates in NX 8 (1) and in NX 8.0.1 (2). Notice that in NX 8.0.1, the software generates a contiguous midsurface along the center of the part, despite the variation in thickness.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active idealized part
Toolbar	**Advanced Simulation® Midsurface by Face Pairs** 

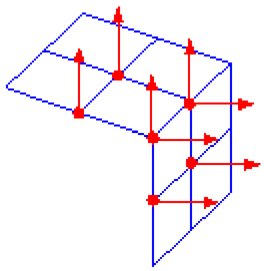
Meshing

Translating elements along their normals

What is it?

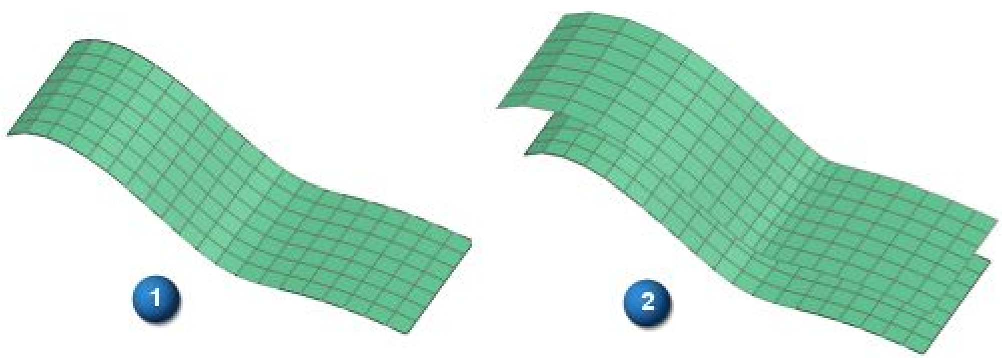
In the **Element Copy and Translate** dialog box, you can use the new **Element Normal** option in the **Direction** list to translate selected elements along their nodal normals.

The software computes the nodal normal of an element based on the normals of the elements connected by a given node:



To determine the projection vector, the software calculates an included angle-weighted average of the normals of all the selected elements. Each element may have a different normal, particularly on non-planar surfaces. When you are translating elements with different normals, the size and shape of the elements may change.

The following example shows a mesh of 2D quadrilateral elements on a slightly curved surface (1). In (2), all of the elements in that mesh are translated 20mm using the **Element Normal** option from the **Direction** list. Notice how the size of some of the translated elements is different.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® Element Copy and Translate**

Menu	**Insert® Element® Element Copy and Translate**

Support for manual node and element commands in axisymmetric models

What is it?

You can now use many of the commands on the **Element Operations** and **Node Operations** toolbars to manually create and modify nodes in an axisymmetric model. In previous releases, you could not use many of these commands when you were working in an axisymmetric environment.

The following tables provide details on the support for each command in axisymmetric models.

Support for Node Operations commands

Command	Supported in axisymmetric models	Notes
Node Create	Yes	You cannot create nodes outside of the axisymmet
Node Translate	Yes	You cannot translate nodes off the axisymmetric p
Node Rotate	Yes	You cannot rotate nodes off the axisymmetric plan
Node Modify Label	Yes	
Node Between Nodes	Yes	You cannot create nodes outside of the axisymmet
Node on Point/Curve/Edge	Yes	You cannot create nodes outside of the axisymmet
Assign Nodal Coordinate System	Yes	
Node Delete	Yes	
Node/Element Information	Yes	
Nodal Coordinate System	Yes	
Node Reflect	Yes	You cannot reflect nodes off the axisymmetric pla
Node Drag	Yes	You cannot drag nodes off the axisymmetric plane
Node Align	Yes	You cannot align nodes off the axisymmetric plan
Node Modify Coordinates	Yes	You cannot modify the coordinates of nodes such axisymmetric plane.

Support for Element Operations commands in an axisymmetric environment

Command	Supported in axisymmetric models	Notes
3D Sweep Between	No	Not currently supported in axisymmetric models.

Command	Supported in axisymmetric models	Notes
Combine Triangles	Yes	
Element Create	Yes	You cannot create nodes outside of the axisymmet
Element Copy and Project	No	**Element Copy and Project** is not currently suppo models.
Element Copy and Reflect	Yes	You cannot reflect elements off the axisymmetric
Element Copy and Translate	Yes	You cannot translate elements off the axisymmetr
Element Delete	Yes	
Element Extract	No	
Element Extrude	No	
Element Modify Associated Data	Yes	
Element Modify Connectivity	Yes	When you modify an element's connectivity, you ca off the axisymmetric plane.
Element Modify Label	Yes	
Element Modify Order	Yes	
Element Revolve	No	
Move Node	Yes	You cannot move an element's nodes off the axisy
Split Shell	Yes	

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with **Axisymmetric Structural** or **Axisymmetric Thermal** as the specified analysis type
Toolbar	**Node Operations** and **Element Operations**

New tolerances for bolted and 1D connections

What is it?

In this release, you can now control several new advanced tolerance values used by the **Bolt Connection** and **1D Connection** commands. In previous releases, these tolerances were internally controlled by the software.

Advanced tolerances for bolt creation

In the **Bolt Connection** dialog box, you can now adjust the following tolerance values:

- **Alignment Tolerance**: Determines whether the head of the bolt and the nut or tapped hole are aligned.

The software finds the vector that is directed from the center of the bolt's head and the center of the bolt's nut or tapped hole. The software then finds a second vector at the center of the bolt's head that is normal to the surface on which the bolt's head is located. The dot product of the two vectors must be within the specified **Alignment Tolerance** for the software to create the bolt.

- **Junction Merge Tolerance**: If your model contains multiple junction planes, this tolerance determines whether the software merges the junction planes. If the distance between the junction planes is less than this value, the software merges the junction planes.

- **Leg Node Tolerance**: Adds an additional radial distance to the specified **Spider Diameter** value, which the software uses to search for candidate leg nodes in spider-type connections.

Advanced tolerance for 1D connections

In the **1D Connection** dialog box, when you select **Edge-to-Face** from the **Type** list, a new **Projected Point Proximity to Face** tolerance is now available.

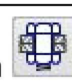
This tolerance determines whether the software uses a projected node as one of the leg nodes in the RBE2 or RBE3 element.

- If the projected node falls outside of the face but within this tolerance value, the software uses the node to create an RBE2 or RBE3 element.

- If the projected node falls outside of the face and outside of this tolerance value, the software does not use the node to create an RBE2 or RBE3 element.

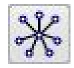
Where do I find it?

Bolt Connection

Application	Advanced Simulation
Prerequisite	An active FEM file with NX Nastran or MSC Nastran as the specified solver
Toolbar	**Advanced Simulation® Bolt Connection** 
Menu	**Insert® Mesh® Bolt Connection**

1D Connection

Application	Advanced Simulation
Prerequisite	An active FEM file

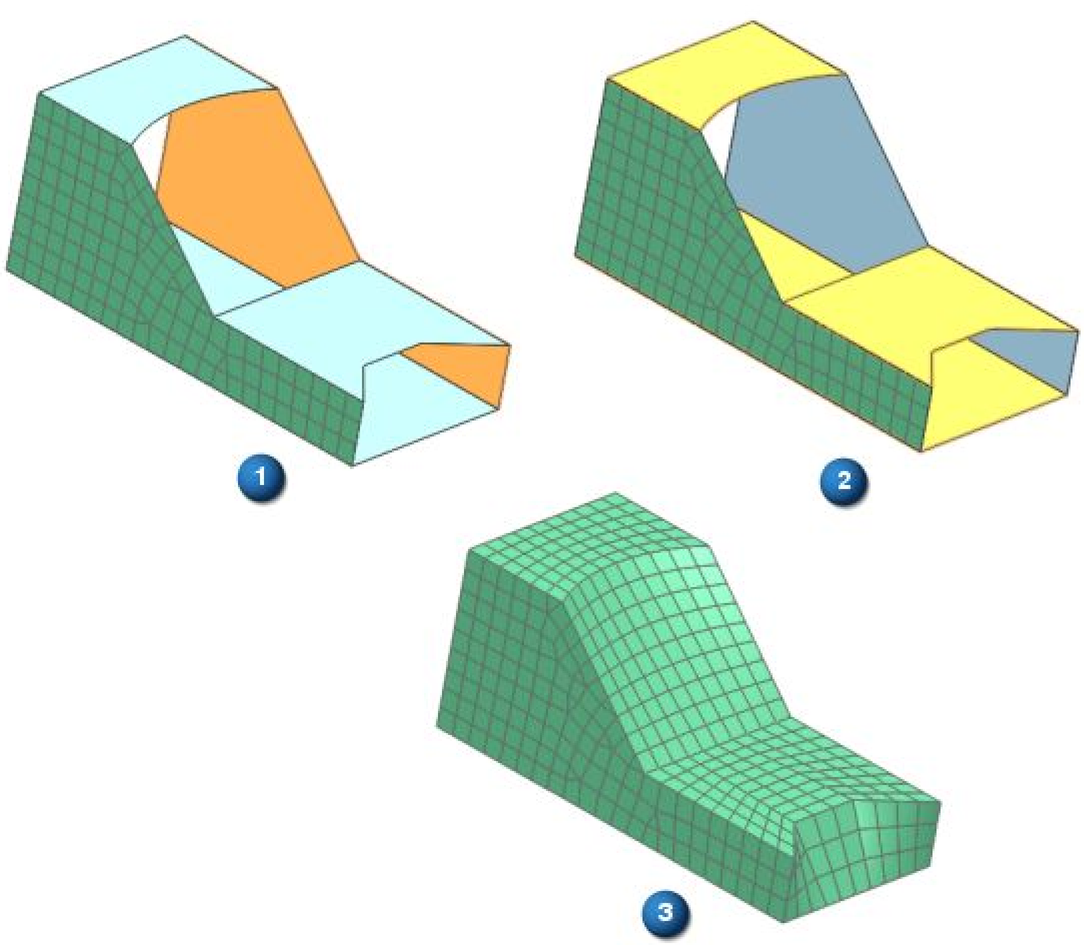
Toolbar	**Advanced Simulation® 1D Connection** 
Menu	**Insert® Mesh® 1D Connection**
Location in dialog box	**Connection Element Creation Tolerance** group

Manual swept meshing now supports wall faces

What is it?

In this release, when you use the **3D Sweep Between** command to define a mesh between a source and a target face, you can now include wall or side faces for the swept mesh to follow. In previous releases, the source and target faces had to be connected in a straight line. This enhancement significantly expands the types of geometry on which you can use the **3D Sweep Between** command to manually sweep a solid mesh.

The following graphic shows a very simple example of the inclusion of wall faces with **3D Sweep Between** . (1) shows the 2D mesh on the source face with the target face highlighted in orange. (2) shows the three selected wall faces highlighted in yellow. (3) shows the resulting hexahedral mesh on the geometry.



Note With the **3D Sweep Between** command, the source and target faces should not be located on the same solid body. If the source and target faces are part of the same solid body, you should use the **3D Swept Mesh** command instead to generate the hexahedral mesh.

Wall faces can be selected manually or automatically

In the **3D Sweep Between** dialog box, you can use the new **Wall Selection** options to select the wall faces manually. You can also click the **Automatic Wall Selection**  button to have the software select the wall faces automatically. If you use the **Automatic Wall Selection** option, the software selects all faces that share edges with both the source and target faces.

You can also use the **Dynamic Wall Selection** option to have the software automatically add all faces that share edges with both the source and target faces to the selected wall faces. With the **Dynamic Wall Selection** option, if you change the specified source or target faces, the software automatically updates the selected wall faces.

Wall faces can be meshed or unmeshed

The faces you select as wall faces can be either meshed or unmeshed. However, if a wall face has an existing 2D mesh, that mesh will determine the number of elements the software sweeps along that wall face to the target face and not the value you specify in the **Number of Layers** box.

Rules for including wall faces in a manual swept mesh

- The source face must be meshed and cannot have a status of **Update Pending**.
- The target face must have the same number of edges as the source face.
- The wall faces must be stitched to both the specified source and target faces.
- You cannot use **3D Sweep Between** to sweep elements between meshes that do not have any associated geometry. You must use the **Face from Mesh** command to create new polygon faces from those elements before you can use the **3D Sweep Between** command.
- You must divide cylindrical faces before you can select them as wall faces. You can use the **Split Face** command to divide the cylindrical faces.

Where do I find it?

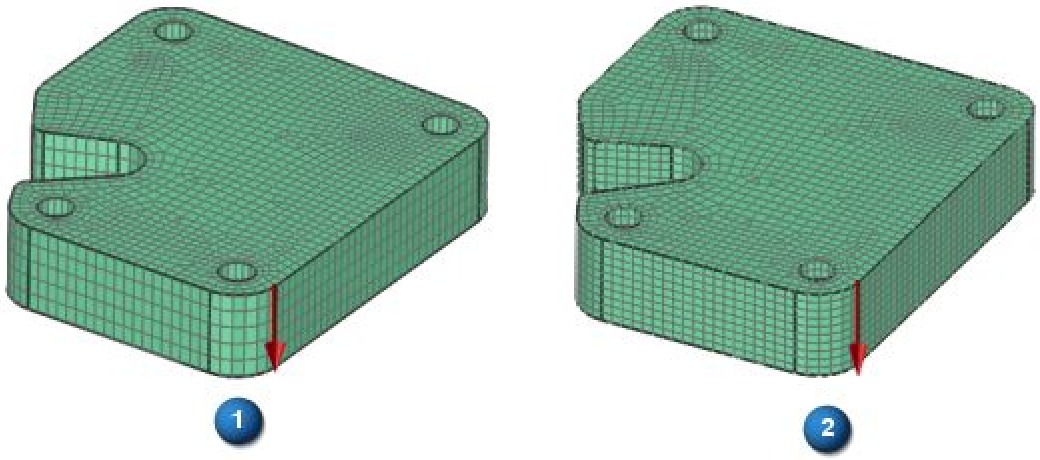
Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations® 3D Sweep Between**
Menu	**Insert® Element® 3D Sweep Between**

Ability to control the number of layers through a volume in a 3D Swept Mesh

What is it?

The new **Use Layers** option in the **3D Swept Mesh** dialog box lets you control the number of layers that the software generates between a source face and a target face. This also allows you to control the size of the elements that the software sweeps through the volume.

In the graphic below, the red arrows indicate the sweep direction from the source face on the top of the volume to the target face on the bottom of the volume. Both meshes shown were generated with the new **Use Layers** option selected. In (1), the specified **Number of Layers** is 5, while in (2), the specified **Number of Layers** is 10.



Where do I find it?

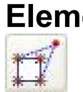
Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 3D Swept Mesh** 
Menu	**Insert→Mesh→3D Swept Mesh**

Element Modify Connectivity support in assembly FEMs

What is it?

You can now use the **Element Modify Connectivity** command to modify the connectivity of any elements that you create in an assembly FEM file. For example, you can use **Element Modify Connectivity** to replace a node from one element with another node. This allows you to connect two separate meshes by specifying where elements should share nodes.

Where do I find it?

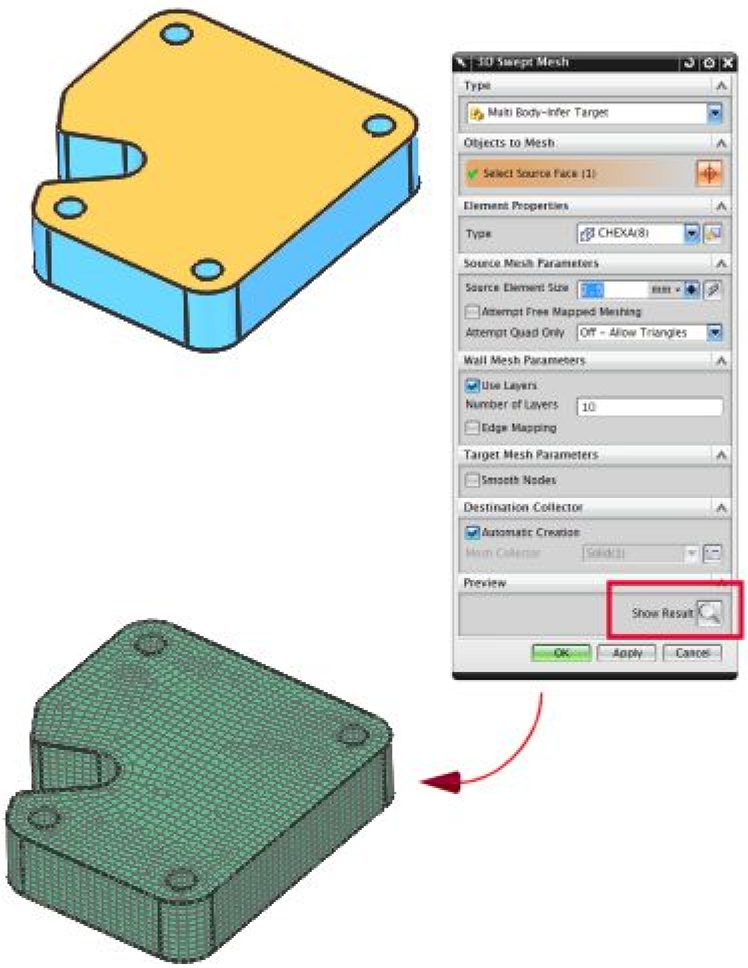
Application	Advanced Simulation
Prerequisite	An active assembly FEM file
Toolbar	**Element Operations® Element Modify Connectivity** 
Menu	**Edit® Element® Modify Connectivity**

Full mesh preview now available in selected commands

What is it?

A new **Show Result** button replaces the **Boundary Nodes** button in the following dialog boxes:

- **2D Mesh**
- **2D Local Remesh**
- **3D Swept Mesh**
- **3D Sweep Between**



You can use **Show Result** to generate a preview of the entire mesh based on your current selections in the dialog box. In previous releases, you could only use the **Boundary Nodes** option to preview the distribution of nodes along the boundaries of your geometry.

After you click **Show Result**, you can:

- Click **Undo Result** if you are not satisfied with the previewed mesh and want to make further adjustments to the settings in the dialog box.
- Click **OK** or **Apply** to accept the previewed mesh.
- Click **Cancel** to exit the dialog box without generating a mesh.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 2D Mesh**  or **2D Local Remesh**  **3D Swept Mesh**  or **3D Sweep Between** 

JT output now available from the Batch Meshing Utility

What is it?

You can now output meshes in JT format files from the NX Batch Meshing Utility . To export a JT file of a mesh, type the following syntax at the command line:

```
ufx_matchmeshing —jt
```

For JT output files, the Batch Meshing Utility uses the following naming convention:

*CAD-name*_MESH_JT_*keyword*.jt

Boundary conditions

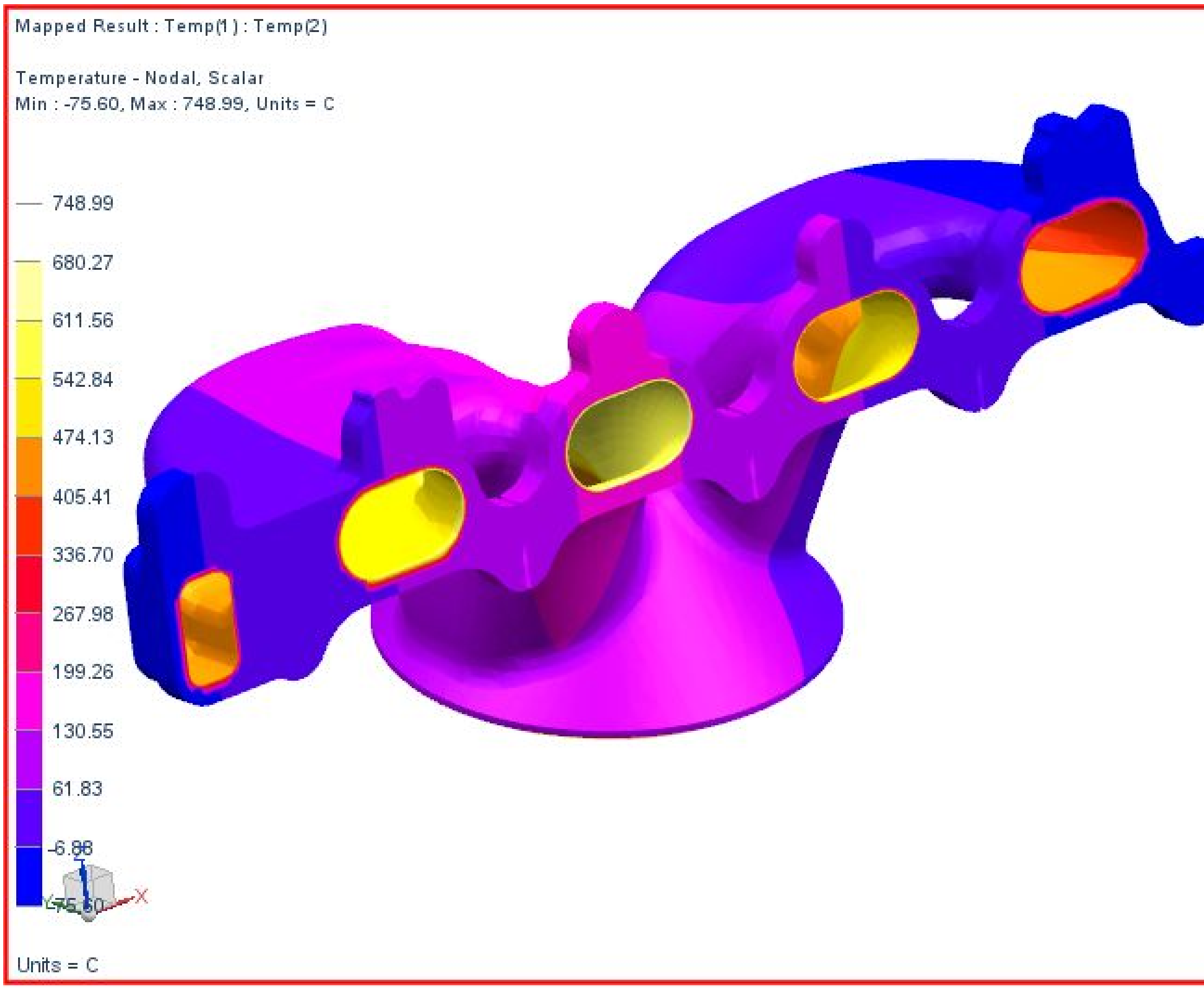
Boundary condition contour display enhancements

What is it?

NX 8.0 introduced the capability to generate a contour plot of pressure, nodal pressure, or temperature loading as a standard post view. With this release, this capability has been expanded to support most types of loads, constraints, and solver-specific simulation objects that contain a value.

Also, you can now create a contour plot of multiple loads and constraints with the **Plot BC Contours** command. When your solution contains multiple boundary conditions, this command opens a dialog box that lists the boundary conditions that are currently included in the solution. You select a boundary condition from the list as the “seed” item. The remaining boundary conditions that are compatible with the seed item are available to be selected. The incompatible boundary conditions are displayed in red and are unavailable.

When you generate the contour plot, the **Post-Processing** toolbar is enabled, and you can use the commands on this toolbar to view and interrogate your loading conditions. You can use the **Edit Post View**  command to modify the loading display, or you can use the **Identify Results**  command to probe loading values at nodes and write them to a spreadsheet or comma-separated-value (CSV) file.



Contour plot of two thermal constraints

Where do I find it?

Application	Advanced Simulation

Prerequisite	A Simulation file containing one or more loads, constraints, and/or solver-specific simulation objects that contain a value
Simulation Navigator	Right-click one or more loads, constraints, or solver-specific simulation objects ® **Plot Contours**
	Right-click the solution node ® **Plot BC Contours**

Nastran support enhancements

Creation of groups during import from included files

What is it?

When you import a Nastran input file that contains included files (external files that are inserted by the `INCLUDE` bulk data entry), NX now creates a group of nodes and elements for each included file.

- NX creates one group for each included file.
- NX creates the groups in the associated FEM file.
- NX names the groups using the following convention: `INCLUDE NameOfTheIncludedFile.Extension`. For example, if your imported Nastran input file uses an `INCLUDE` command to insert a file named `BACKSHELL_DH.blk`, NX creates the following group in the **Simulation Navigator**:

  ```
  INCLUDE BACKSHELL_DH.blk
  ```

Group creation for nested included files

In some cases, the Nastran input file that you import may contain nested included files. Nested included files occur when the external file or files that you insert with the `INCLUDE` command also contain included files. If your input file contains nested included files, NX creates a separate group for each included file.

For example, if you import a file named `input.dat` that uses an `INCLUDE` command to include the file `level2.dat`, which:

- Contains grid points (nodes) 1 to 5.
- Uses an `INCLUDE` command to include the file `level3.dat`.

  The file `level3.dat` contains grid points 6 to 10.

In this example, when you import the `input.dat` file, NX creates the following groups in the **Simulation Navigator**:

```
INCLUDE level2.dat
INCLUDE level3.dat
```

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**

Import defaults for CONM2 and CELAS mass and stiffness values

What is it?

This release includes new **Element Import Options** customer defaults that you can use to control whether NX imports mass and stiffness values for Nastran CONM2 and CELAS elements as **Element Associated Data** or **Mesh Associated Data**. Currently, NX imports all CONM2 and CELAS elements that share common mesh associated data into a single mesh within a single mesh collector. NX then stores any specified mass values for CONM2 elements or and stiffness values for CELAS element as **Element Associated Data**.

- You can significantly improve performance by importing a model that contains numerous CONM2 and CELEAS elements with different mass or stiffness values as **Element Associated Data**.

  For example, if you import a model that contains many CONM2 elements, each with a different mass, NX places all the CONM2 elements into a single mesh in the **Simulation Navigator**. However, if you later need to change the mass of a number of those CONM2 elements, modifying many separate **Element Associated Data** values can be cumbersome.

- You will decrease import performance by importing a model that contains numerous CONM2 and CELAS elements with different mass or stiffness values as **Mesh Associated Data**

  For example, if you import a model that contains many CELAS2 elements, each with a different stiffness value, NX places each CELAS2 element an individual mesh in the **Simulation Navigator**. NX places CELAS2 elements that have the same stiffness value in the same mesh. If you later need to change those stiffness values, you can simply modify the **Mesh Associated Data** for the mesh.

Where do I find it?

Application	Advanced Simulation
Prerequisite	**File→Utilities→Customer Defaults**
Location in dialog box	**Simulation→NASTRAN→Solution** tab

Support for new composite stress and strain output options

What is it?

In a laminates analysis, you can use the new **Composite Solid Ply Output** option on the **Stress** or **Strain** tabs in the **Structural Output Requests** dialog box to specify the location at which you want the software to report element stresses or strains.

The **Composite Solid Ply Output** option lets you control the location of stress or strain output for `CHEXA` and `CPENTA` type elements whose physical properties are defined with a **Solid Laminate** (`PCOMPS`) type of physical property.

You can select:

- **CPLYMID** to request the stresses or strains at the middle of each ply.
- **CPLYBT** to request the stresses or strains at the bottom and the top of each ply.
- **CPLYBMT** to request the stresses or strains at the bottom, middle, and top of each ply.

For more information, see the *STRESS* and *STRAIN* case control commands in the *NX Nastran Quick Reference Guide*.

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
Toolbar	**Advanced Simulation® Modeling Objects**  ® **Structural Output Requests**
Menu	**File® Import® Simulation**

New default value for the memory keyword

What is it?

The default value for the **memory** option in the **Solver Parameters** dialog box has been changed from `estimate` to a blank value. The **memory** option corresponds to the Nastran `memory` keyword.

In Nastran, the `memory` keyword specifies the amount of open core memory to allocate to the solve. If you set `memory` to `estimate`, Nastran runs the `ESTIMATE` utility to estimate the memory and disk requirements for the input file. However, although the `ESTIMATE` utility produces reasonably accurate estimates for certain types of solutions, such as SOL 101 and SOL 103, it does not provide accurate estimates for other solution types, such as contact or SOL 601 analyses.

Changing the default for the **memory** option in NX from `estimate` to a blank value allows you to specify you own estimate for the solve's memory requirements. It also allows you to specify the `estimate` option for those types of solutions where the `ESTIMATE` utility is most accurate.

For more information about memory management in Nastran, see *Determining Resource Requirements* in the *NX Nastran*

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file active with NX Nastran or MSC Nastran as the specified solver
Simulation Navigator	Right-click the active solution® **Edit Solver Parameters**® **General** tab

Model Setup Check now verifies existence of external OP2 files

What is it?

For models that use external superelement reductions, the **Model Setup Check** command now verifies the existence of any associated OP2 files before you solve. Within an assembly FEM file, the representation of external superelement data is defined with the full path to the OP2 file. If the location of that OP2 data is later moved or deleted, the external superelement definition is lost.

Beginning in this release, when you solve a Nastran model using the **Model Setup Check** option or export a Nastran solution, NX now issues an error alerting you that it cannot locate the necessary OP2 file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file active with NX Nastran or MSC Nastran as the specified solver
Simulation Navigator	Right-click the active solution® **Solve**® **Model Setup Check**

Abaqus support enhancements

Abaqus 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
`*CONTROL`	All parameters are supported for import and export.	Yes	Yes	In NX 8, the `*CONTROL` keyword was only supported for export.
`*NODE FILE/*EL FILE/*CONTACT FILE`	All parameters are supported for import and export.	Yes	Yes	In previous releases, these .fil output file request keywords were only partially supported for import.
`*NODE PRINT/*EL PRINT/*CONTACT PRINT`	All parameters are supported for import and export.	Yes	Yes	In previous releases, these .dat output file request keywords were only partially supported for import.
`*OUTPUT/*NODE OUTPUT/*ELEMENT OUTPUT/*CONTACT OUTPUT`	All parameters are supported for import and export.	Yes	Yes	In previous releases, these .odb output file request keywords were not supported for import.

Keyword	Supported parameters	Import support	Export support	Notes
*PRINT	All parameters are supported for import and export.	Yes	Yes	In previous releases, this .msg/.sta output file request keyword was not supported for import.
*RADIATE	All parameters are supported for import and export.	Yes	Yes	You can now import the *RADIATE keyword into the axisymmetric thermal environment.

New option for locking bolts after the application of a pre-load

What is it?

In a pre-loaded bolt analysis, you can use the new **Lock Bolts After Pre-Load Application** option to control the behavior of bolts in the steps after you remove the pre-load.

- If you select the **Lock Bolts After Pre-Load Application** check box, in all steps after you remove a **Bolt Pre-Load**, the software removes the bolt load and constrains the length of the bolt to its pre-tensioned length
- If you clear the **Lock Bolts After Pre-Load Application** check box, the software removes the bolt load but does not maintain the pre-tensioned length of the bolt.

For example, consider a solution that consists of four steps. The **Bolt Pre-Load** is applied only in the first step. If you select **Lock Bolts After Pre-Load Application** in the **Solution Options** dialog box, when you solve the model, NX applies the pre-load only in the first step and calculates the length of the bolt. Then, in steps 2–4, NX fixes the length of the bolt to the length calculated in step 1.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file active with Abaqus as the specified solver

Simulation Navigator	Right-click the Simulation file ® **New Solution**® **General** tab

ANSYS support enhancements

Support for nonlinear buckling analyses

What is it?

This release includes support for ANSYS nonlinear buckling solutions.

- A linear or *eigenvalue* buckling analysis predicts the theoretical buckling strength of an ideal linear elastic structure. A linear buckling analysis computes the structural eigenvalues for a given system. In a linear buckling analysis, the software applies perturbations to the undeformed geometry and looks for deflections that could be unstable under a specific set of loads.

- A nonlinear buckling analysis predicts the actual response of a structure for each load increment. A nonlinear buckling analysis is more accurate than a linear buckling analysis because it uses nonlinear, large deflection static analysis to predict the buckling loads.

Solution step types

In Advanced Simulation, when you create a **Nonlinear Buckling** type of solution, you can create the following types of analysis steps:

- **Linear Buckling**, in which you perform an initial linear (eigenvalue) buckling analysis to determine the critical loads that induce buckling and possible buckling modes. In this type of step, you can analyze the effects of initial imperfections as well as shape multipliers. After you solve a **Linear Buckling** step, you can

  - View the buckling mode shapes and load multipliers.

  - Look at the results to find the iteration time step in which the structure becomes unstable. If that occurs, this time step becomes the starting point for the **Nonlinear Buckling** step.

- **Nonlinear Buckling**, in which you solve the model with large deflection active (`NLGEOM, ON`). In this step, the model is stressed to reach its limit or maximum load.

- **Post-Buckling**, which is a continuation of a **Nonlinear Buckling** step. After a load reaches its buckling value, the load value may remain unchanged or may decrease while the deformation continues to increase.

Support for nonlinear stabilization

This release also includes support for nonlinear stabilization during a **Post-Buckling** step. Nonlinear stabilization is a tool for managing both local and global instabilities. In ANSYS, nonlinear stabilization consists of adding an artificial damper or *dashpot* element at each node of an element that supports the technique.

- Before buckling occurs, the system may have low displacements over a given time step. Conceptually, you can think of this as a low *pseudo velocity* that does not generate much resistive force from the dampers.

- When the buckling occurs, the system may have larger displacements over a small time step. As a result, the *pseudo velocity* becomes large and does generate a large resistive force from the dampers.

NX now supports the use of the ANSYS `STABILIZE` command to turn nonlinear stabilization either on or off for a particular **Post Buckling** step. You can specify nonlinear stabilization either by specifying a damping factor or an energy dissipation ratio.

Ability to preview the ANSYS monitor file

A new **Preview Ansys monitor file** button has been added to the **Solution** dialog box. After you solve a solution, you can use the **Preview Ansys monitor file** button to examine a summary of your results (the convergence table). You can then determine which sub-step (the sub-step where divergence occurs) to use in the **Post-Buckling** step.

Newly supported ANSYS commands

This release now includes import and export support for the following ANSYS commands that are necessary for a nonlinear buckling analysis:

ANSYS Command	Description
`PRED`	Activates a predictor
`RESCONTROL`	Controls the writing of multiframe restart files by the solver.
`STABILIZE`	Controls nonlinear stabilization
`UPGEOM`	Adds displacements from a previous analysis and updates the finite element model to the deformed configuration.

Where do I find it?

Application	Advanced Simulation

Prerequisite	An active Simulation file with ANSYS as the specified solver, **Structural** as the specified **Analysis Type** and **Nonlinear Buckling** as the specified **Solution Type**
Simulation Navigator	Right-click a Simulation file **New Solution**® **Nonlinear Buckling**

Support for COMBIN39 elements

What is it?

ANSYS COMBIN39 elements are now supported in Advanced Simulation. A COMBIN39 is a unidirectional nonlinear spring element with a nonlinear, generalized force-deflection capability. COMBIN39 elements:

- Can be used in any type of structural analysis.
- Have a large displacement capability for which there can be two or three DOF at each node.
- Have no mass of thermal capacitance.

In NX, you can create COMBIN39 elements from the **1D Mesh** dialog box. You can also import ANSYS input files that contain COMBIN39 elements.

For more information about COMBIN39 elements, see:

- *COMBIN39* in the *ANSYS Elements Reference* guide.
- *COMBIN39-Nonlinear Spring* in the *ANSYS Theory Reference* guide.

Support for COMBIN39 Real Constants and KEYOPTS

In ANSYS, you can define additional properties for COMBIN39 elements with real constants and KEYOPTS. In NX, you define these additional properties with:

- The new COMBIN39 ET type of modeling object, which lets you set the KEYOPTS for the element. For example, you can specify:
  - The unloading path.
  - The behavior of the element under a compressive load.
  - The type of output for the element.
- The new COMBIN39 physical property table, which lets you set the real constants for the element. With the COMBIN39 element, you can specify a field to define the force-deflection (loading) curve.
  - The maximum number of deflection points you can use to define the curve is 20.

- o You should define the curve so that the deflections increase from the third (compression) to the first (tension) quadrants.
- o Adjacent deflection points should not be closer than 1E-7 times the total input deflection range.
- o The last deflection point should be positive.

Post-processing support

When you solve a model that contains COMBIN39 type elements, you can now view results for those elements in Post-processsing.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with ANSYS as the specified solver
Menu	**Insert® Mesh® 1D Mesh**
Toolbar	**Advanced Simulation® 1D Mesh** 

Thermal element support enhancements

What is it?

This release contains enhancements to how you define thickness properties for the following types of ANSYS thermal elements:

- PLANE55 and PLANE77 elements
- SURF151 and SURF152 elements

PLANE55 and PLANE77 elements

When you use the **Mesh Associated Data** dialog box to define the thickness source for PLANE55 and PLANE77 elements, you can now use a **Physical Property Table** definition or a field to define the thickness data. In previous releases, you could only use a **Physical Property Table** definition.

Note The software only considers the thickness data if you also select **(3) Plane with Z-depth, specified via real constant THK** from the **KEYOPT(3)** list in the **PLANE55 ET** or **PLANE77 ET** dialog box.

SURF151 and SURF152 elements

When you create a **SURF151 Real Constants** or **SURF152 Real Constants** modeling object, you can use the new **Inherit Nodal Thickness from Underlying Elements** option to allow the surface effect elements to inherit their thickness values from the underlying 2D or 3D thermal elements.

For example, consider a model with SURF151 elements defined on the surfaces of PLANE55 elements. In the **Mesh Associated Data** dialog box for the PLANE55 elements, the specified **Thickness Source** is a field. In the **SURF151 Real Constants** dialog box, if you selected the **Inherit Nodal Thickness from Underlying Elements** option, the software would propagate the field thickness data from the PLANE55 elements to the SURF151 elements.

Where do I find it?

PLANE55 and PLANE77 Mesh Associated Data

Application	Advanced Simulation
Prerequisite	An active FEM file with ANSYS as the specified solver and **Thermal** as the specified analysis type
Simulation Navigator	Expand the appropriate mesh collector, then right-click a mesh® **Edit Mesh Associated Data**

SURF151 and SURF152 Real Constants

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with ANSYS as the specified solver and **Thermal** as the specified analysis type
Toolbar	**Advanced Simulation**® **Modeling Objects**  ® **SURF151 Real Constants** or **SURF152 Real Constants**

High performance computing options

What is it?

You can use new options in the **Solver Parameters** dialog box to set high performance computing options. You can now:

- Control the use of shared-memory parallel processors.
- Use GPU acceleration.

Note You must have an ANSYS high performing computing license to solve your model using these options. NX does not check for the existence of a high performing computing license before launching the solve.

Shared-memory parallel processing

In ANSYS, you can shared-memory processing to solve an ANSYS solution across multiple processors on a single machine. Running ANSYS on a shared memory architecture is one of the easiest ways to improve the performance of ANSYS.

In Advanced Simulation, you can now select the new **Use Shared-Memory Parallel** option and then specify the number of processors (`-np`) that you want to use on a single machine.

For more information, see *Using Shared-Memory ANSYS* in the ANSYS *Advanced Analysis Techniques Guide*.

GPU acceleration

ANSYS 13 and higher supports the use of a graphics processing unit (GPU) to increase computing capacity. The ANSYS GPU accelerator works by off-loading some of the most numeric-intensive algorithms from the CPU onto the GPU. Currently, ANSYS only supports the NVIDIA Tesla GPU for use when accelerating ANSYS solutions.

In Advanced Simulation, you can select the new **Use GPU Accelerator** option to use the GPU accelerator capability.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Simulation Navigator	Right-click a solution® **Edit Solver Parameters**

Changes to the handling of user defined text

What is it?

The **User Defined Text** capability from previous releases has been redesigned and enhanced. You can now:

- Insert a **User Defined Text** modeling object into a specified section in your ANSYS input file.
- Remove commands generated by NX when a **User Defined Text** object is inserted into an input file.
- Preview the placement of the text within the input file before solving the model.

Inserting user defined text into a specific section of your input file

Within a solution or solution step, you can now control the exact location in your input file at which the software inserts the **User Defined Text** modeling object. For example:

- At the solution level, you can insert **User Defined Text** into a variety of locations, including before or after the input file header information, or before or after the `ASSIGN` or `/SOLUTION` statement.

- At the step level, depending on your analysis type, you can insert **User Defined Text** into the loads or constraints section of the input file as well as at the beginning or end of the step.

Previewing the placement of user defined text

A new **Preview Input File** button is now available in the **Solution** dialog box. When you click **Preview Input File**, the software generates an abbreviated version of the input file based on your current selections. You can review this previewed file to quickly validate that the placement of the **User Defined Text** within the input file is correct.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Simulation Navigator	Right-click a solution or step® **Edit**

Improved export and solve options

What is it?

In the ANSYS solver environment, this release includes additional options when you export and solve your model. You can now:

- Selectively exclude ANSYS commands from being exported to the input file.

- Access a number of advanced options when you solve an ANSYS solution.

Selective export of ANSYS commands

You now have more granular control over the ANSYS commands that NX exports. You can use the new **Filter Method** list in the **Output Options** group in the **Export Simulation** dialog box to control the ANSYS syntax that NX includes in the ANSYS input file. From the **Filter Method** list, you can select:

- The **By card name** option to exclude specific commands from the export.

- The **By card family** option to exclude categories of commands, such as loads, materials, or the NX header information, from the export.

Access to advanced options when you solve

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 ANSYS input file. For example, you can:

- Control the specific commands that NX includes in the ANSYS input file.
- Specify offsets for the different IDs in the input file, such as nodes, elements, or real constants.
- 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 **Output Options**, the software does not perform a **Model Setup Check** before the solve begins.

Where do I find it?

Selective export options

Application	Advanced Simulation
Prerequisite	A Simulation file active with ANSYS as the specified solver
Menu	**File**® **Export**® **Simulation**

Advanced options available when you solve

Application	Advanced Simulation
Prerequisite	A Simulation file active with ANSYS as the specified solver
Menu	**Analysis**® **Solve**
Simulation Navigator	Right-click a solution ® **Solve**

Post-processing

Improvements to memory management in animation

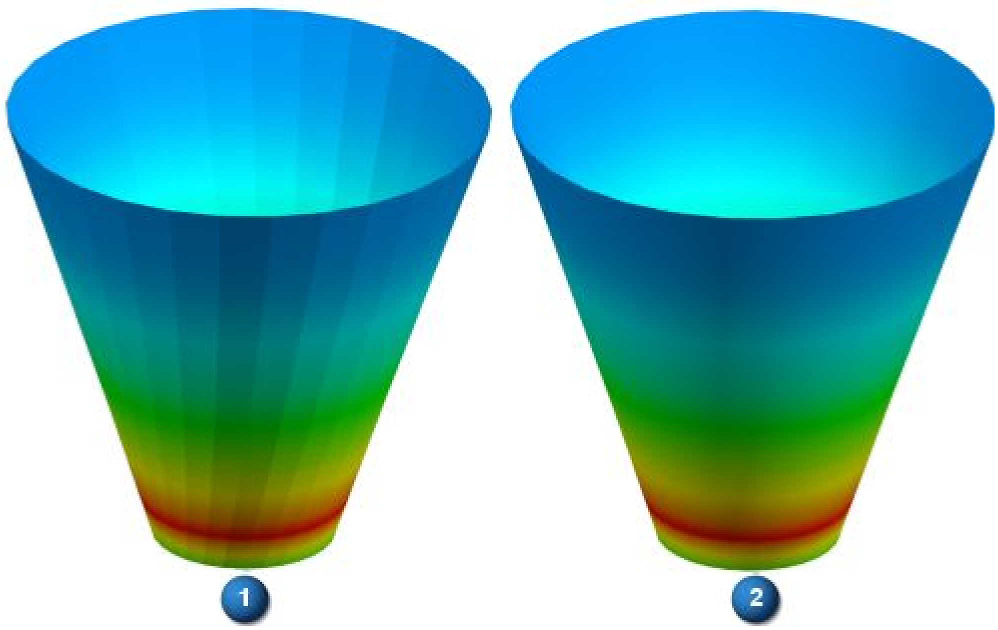
What is it?

Memory management in post-processing **Animation** is improved. In previous releases, animating very large models consumed excessive system memory. New options on the **Post View** dialog box help you to better balance memory consumption and performance when animating large models.

Display Compression

The **Compress** option compresses **Smooth**, **Banded**, and **Element** post view displays and reduces the memory that the post view consumes. When compressed, the shading will be smoothed at edges that have curvature changes.

The **Face Normal Tolerance** and **Value Level Tolerance** options provide additional compression and memory savings, especially for models that have many features or a large variation in results.



Element-nodal stress results: (1) Without Display Compression; (2) With Display Compression

Unlighted display

When you select the **Compress** option, the initial loading of animation frames will be slower. To improve the loading performance of compressed animation frames and to further reduce post view memory consumption, you can clear the **Lighted** check box to turn off the NX ambient lighting. For more information, see Ability to turn off ambient lighting in contour display.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	A solved model with results
Toolbar	**Post-Processing**® **Edit Post View** 
Location in dialog box	**Display** tab® **Compress** and **Lighted** check boxes

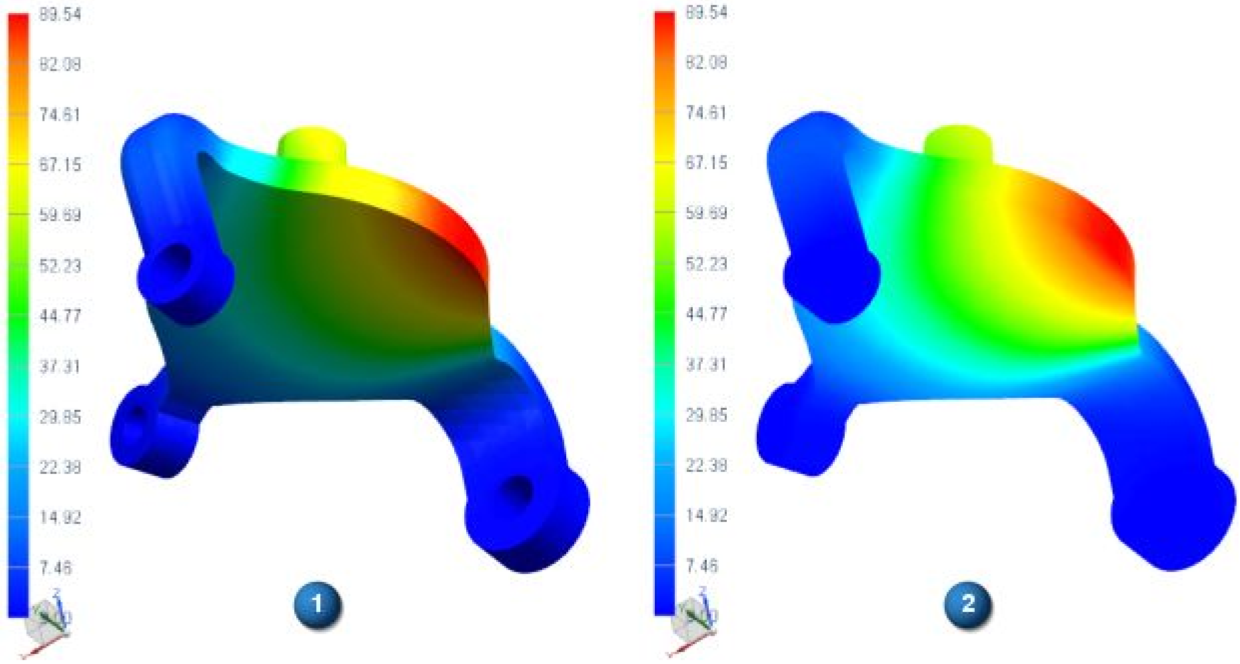
Ability to turn off ambient lighting in contour display

What is it?

You can now display the contours in post processing results without the effects of shadows, by turning off the new **Lighted** option.

Also, when the **Compress** post view option is selected, turning off the **Lighted** option reduces the post view memory consumption, and improves the performance of loading the frames of an animation.

The following examples show displacement results.



(1) Lighted turned on; (2) Lighted turned off

In example (1), notice that the shadows help you see the model features more clearly, but they also obscure the contour colors. In example (2), without the shadows, the contour colors match the colors in the color bar legend. However, the shape of the geometry is not as clear, and features are not as visible.

The above example is shown with external element edges hidden to better illustrate the color differences. However, the display of external element edges can help you distinguish model features when the **Lighted** option is turned off.

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	A solved model with results
Toolbar	**Post-Processing® Edit Post View** 

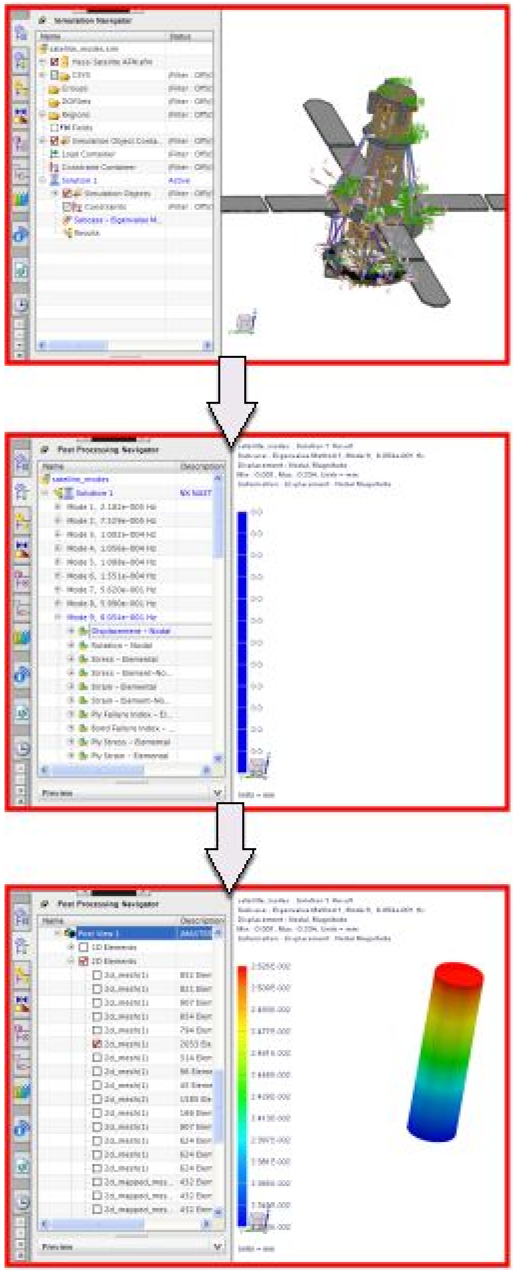
Location in dialog box	**Display** tab® **Lighted** check box

Create an empty post view

What is it?

You can now display a post view with all meshes initially hidden, using the new **Empty Post View** command. You can then selectively display the meshes as needed. This option is useful for when you want to display very large models that contain multiple meshes, in which the initial display of the results takes considerable time. It is also useful for displaying only a region of interest.

After you select the **Empty Post View** option, specify your results. In the **Post-Processing Navigator**, under the appropriate post view, select the meshes and elements for which to display results. You can edit the post view regardless of whether any meshes are displayed.

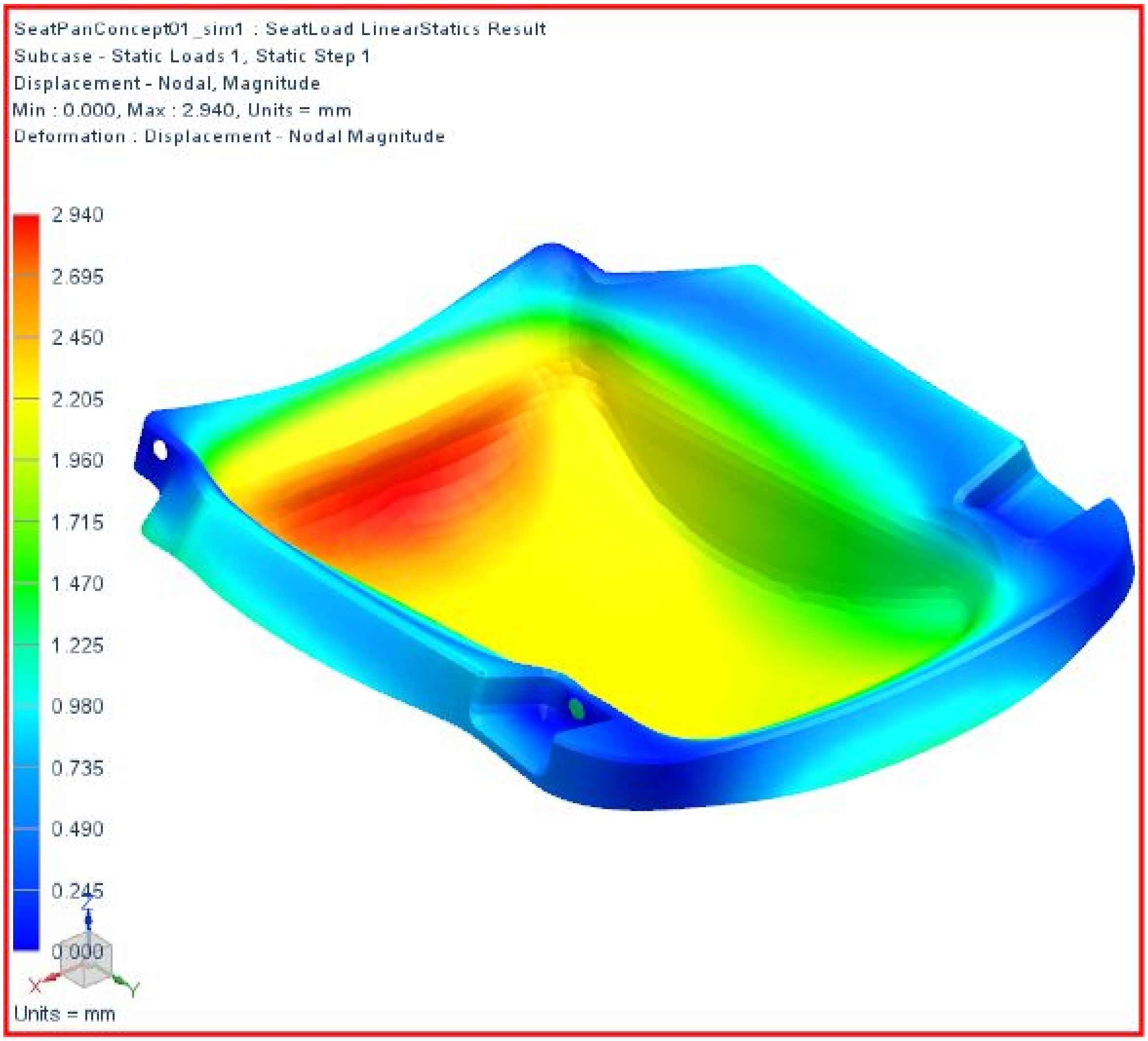


Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved model with results
Toolbar	**Post Processing® Empty Post View** 

Element edge display customer default for post view

A new customer default lets you set the default setting of the **Primary Display Edges** list. Setting the element edge display to **None** can improve post view display performance and memory consumption considerably, and can make the analysis results easier to interpret.



Where do I find it?

Customer Default

Application	Advanced Simulation, Design Simulation
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Customer Defaults** dialog box® **Simulation**® **Post Processor**® **Post View** tab® **Primary Display** group® **Edges** list

Command

Application	Advanced Simulation, Design Simulation

Toolbar	**Post Processing**® **Edit Post View** ® **Edges & Faces** tab® **Primary Display** group® **Edges** list

Improved absolute value calculation for derived components

What is it?

In previous releases, when you selected the **Absolute Value** option for results derived from tensor components, NX calculated the derived results using the absolute values of the tensor components.

Beginning with this release, the **Derived Component** option is now available when you select the **Absolute Value** option. The **Derived Component** option specifies that NX first calculates the derived quantity using the unaltered values of the tensor components, and then takes the absolute value of the derived result, such that reported results are all algebraically positive.

Derived Component is the default option. The old calculation method, now named the **Tensor Components** option, remains available for legacy purposes.

The **Derived Component** option is available for these result quantities:

- Determinant
- Mean
- Max Shear
- Min, Mid, and Max Principal
- Octahedral

Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved model with tensor-derived results
Toolbar	**Post Processing**® **Set Result**
Location in dialog box	**Absolute Value** check box® **Derived Component**

Improvements to exporting identified results

Exporting deformed coordinates to a CSV file

When you export results selected with the **Identify** command, you now can export the deformed coordinates associated with the result. The **Deformation** settings defined for the post view will be used for the exported results. For example, if you export nodal von Mises stress, the deformed coordinates of the corresponding nodes are also exported.

Specifying delimiters for exported results

When you export results selected with the **Identify** command, you can now specify any delimiter to separate the data fields. This option is useful for importing the results into other applications that require a specific delimiter.

```
1  Node ID      X Coord      Y Coord      Z Coord            X            Y
     20243 -1.9000e+002  1.2700e+003 -1.2500e+001 -5.6676e-002 -1.3834e-002

2  Node ID*     X Coord*     Y Coord*     Z Coord*           X*           Y*
     20243*-1.9000e+002* 1.2700e+003*-1.2500e+001*-5.6676e-002*-1.3834e-002*
```

(1) Default delimiter of 2 spaces; (2) Asterisk delimiter

Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	A solved model with results
Toolbar	**Post Processing**® **Identify** ® **Export Selection to a File**
Location in dialog box	**Write Deformed Coordinates** option **Delimiter** option

Expanded results data set support in Post-Processing

What is it?

You can now display and analyze results in post-processing for these additional result types.

- Nastran **Solid Laminate Failure Index** and **Solid Inter-Laminar Failure Index** results. For more information, see Solid ply results in laminate post processing.

- Nastran **External Applied Heat Flow** results, generated using the **Applied Load** output request (OLOAD, SORT1) in NLTCSH 153 or NLTCSH 159 solutions.

- Results from ANSYS COMBIN39 elements. For more information, see Support for COMBIN39 elements.

Component nomenclature for tensor results in Native coordinate system

What is it?

When you display tensor results using the **Native Coordinate System** option in the **Set Result**, **Cutting Plane**, or **Free Body Results** dialog boxes, the stress and strain components are now labeled as 11, 22, 33, 12, 23, and 31 instead of XX, YY, ZZ, XY, YZ, and ZX.

This change better supports the interpretation of ply stress and strain results. For more information, see Ply stress and strain label nomenclature.

Response Simulation

Improvement to Obsolete Check Status command

What is it?

In previous releases, when you re-solved the SEMODES 103 – Response Simulation modal solution after creating a response simulation event and excitations, the response simulation became obsolete and unusable. You could use the **Check Obsolete Status** command to clear the obsolete status, but only if you had not changed the modal settings, such as damping and the list of active modes to be used for the response simulation.

Beginning with this release, a new customer default, **Allow Override of Obsolete Status**, allows you to use the **Check Obsolete Status** command to clear the obsolete status of your response simulation, regardless of whether its modal settings are consistent with the modal solution.

After clearing the obsolete status, you must make sure the response simulation is still consistent with the modal solution.

Where do I find it?

Customer default

Application	Advanced Simulation
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Customer Defaults** dialog box® **Simulation**® **Response Simulation**® **Environment** tab® **Allow Override of Obsolete Status**

Command

Application	Advanced Simulation
Prerequisite	An NX Nastran SEMODES 103 – Response Simulation solution and a response simulation solution process
Simulation Navigator	Right-click the response simulation solution process® **Check Obsolete Status**

Ability to unlock modal settings

What is it?

After you generate response results, the modal settings, such as damping and list of active modes, are locked. A new command, **Unlock Modal Settings**, lets you make changes to these settings after you generate response results.

This command is useful when you want to compare two sets of modal settings. For example, you can clone an event, change the modal settings of the response simulation, and then generate a new set of response results to compare with the original results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An NX Nastran SEMODES 103 – Response Simulation solution and a response simulation solution process
Simulation Navigator	Under **Modal Representation**, right-click **Normal Modes**, **Constraint Modes**, or **Attachment Modes**® **Unlock Modal Settings**

Ability to clone obsolete response simulations and events

What is it?

You can now clone response simulation solution processes or individual events whose status is **Obsolete**. In previous releases, you could clone only response simulation or events whose status was **Active**.

You can also create a copy of an event by dragging it from one response simulation to another.

When you clone or copy an event, the excitations are copied along with the event.

When you clone a response simulation, all events, excitations, sensors, and strain gages are copied.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An NX Nastran SEMODES 103 – Response Simulation solution and a response simulation solution process
Simulation Navigator	Right-click the response simulation solution process or event® **Clone**

Improvement to file names of copied sensors and strain gages

What is it?

When you copy a sensor or strain gage from one response simulation to another, the file name of the new sensor or strain gage will be appended with an incrementing number if a sensor or strain gage already exists with the same name. In previous releases, the words "copy_of_" were prefixed to the new file name. This prefix is no longer applied.

For example, when you copy a sensor named `my_sensor_1` from response simulation "A" to response simulation "B," the new sensor will be named `my_sensor_1`. If you were then to copy the same sensor again, from response simulation A to response simulation B, the name of the new sensor would be `my_sensor_1_001`.

NX Laminate Composites

Solid ply results in laminate post processing

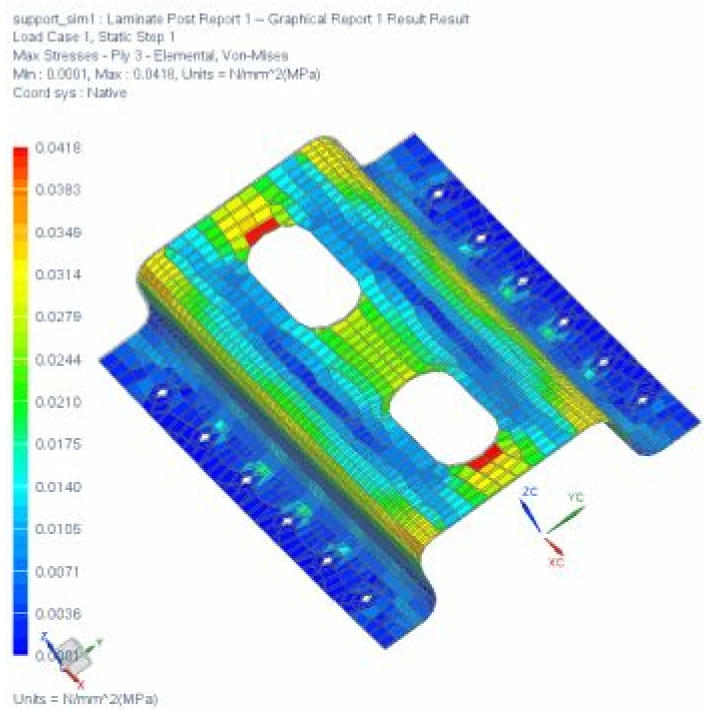
What is it?

Laminate advanced post processing can now use the solid ply results that you calculate using the Nastran solver's PCOMPS card.

You can:

- Export the results to a spreadsheet or a CSV file using the **Spreadsheet Report** command.
- View the results graphically using the **Graphical Report** command.
- Export the results without querying the active FEM using the **Quick Report** command.

You can obtain ply stress, ply strain, ply failure index, ply margin of safety, and ply strength ratio results.



Note You cannot obtain bond failure index, bond margin or safety, or bond strength ratio results for solid laminates.

Where do I find it?

Application	Advanced Simulation
Prerequisite	The laminate post report must point to a solution that includes solid ply results.
Simulation Navigator	Right-click the laminate post report node ® **Create Spreadsheet Report** or **Create Graphical Report** or **Create Quick Report**

Import Layup enhancements

What is it?

When you use the **Import Layup** command, you can now import:

- FiberSIM ply-based layups from a FiberSIM HDF5 (*.h5) file. In previous releases, you could only import layups from a FiberSIM XML file.

  In addition to the information contained in a FiberSIM XML file, the FiberSIM HDF5 file also contains, for each ply:

  - o The starting point of the draping
  - o The direction of the draping

   - o The orientation angle

- Layups on selected elements. In previous releases, you could only select polygon faces.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Laminates ® Import Layup**
Menu	**Insert ® Laminate ® Import Layup**

AFEM support enhancements for NX Laminate Composites

What is it?

- You can now use the advanced laminate post report tools to post process ply results from a model in an assembly FEM (.afm).

- Global layup plies are now correctly exported to structural solvers from an assembly FEM (.afm). This includes imported and inflated layups.

Note Zones are calculated in the FEM and not in the assembly FEM. If you rotate the assembly's FEM instance, the zone ply angles are calculated using incorrect reference because the zones are still calculated using the material orientation vectors from the FEM.

Export All Design Variables as Modeling Objects

What is it?

Use the new **Export All Design Variables as Modeling Objects** option to export all continuous ply angle and ply thickness design variables.

In previous releases, you needed to export each design variable one by one using the **Export Selected Design Variable as Modeling Object** option.

Why should I use it?

This option is useful when you work with models with a large number of ply angle and ply thickness design variables. You can now easily export all continuous design variables as **Design Variable — Composite Property** modeling objects to use in a Nastran DESOPT 200 solution.

Where do I find it?

Application	Advanced Simulation

Prerequisite	You must work in the FEM.
Toolbar	**Laminates** ® **Laminate Physical Property**
Menu	**Insert** ® **Laminate** ® **Laminate Physical Property**
Location in dialog box	Select the **Enable Optimization** check box ® **Export All Design Variables as Modeling Objects**

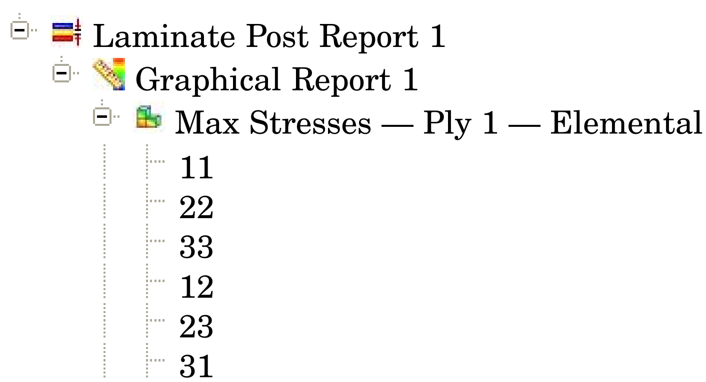
Ply stress and strain label nomenclature

What is it?

The labels for ply stress and strain components are changed in the **Post Processing Navigator**. Because ply stress and strain results are always computed in the native coordinate system, XX, YY, ZZ, XY, YZ, and ZX are replaced with 11, 22, 33, 12, 23, and 31. The native coordinate system for ply results is the ply coordinate system.

In ply stress or strain results, component:

- **11** = Ply longitudinal stress or strain
- **22** = Ply transverse stress or strain
- **33** = Cross-ply stress or strain
- **12** = Ply in-plane shear stress or strain
- **23** = Ply out-of-plane shear stress or strain
- **31** = Ply out-of-plane shear stress or strain



Durability

Evaluate Fatigue Histories

What is it?

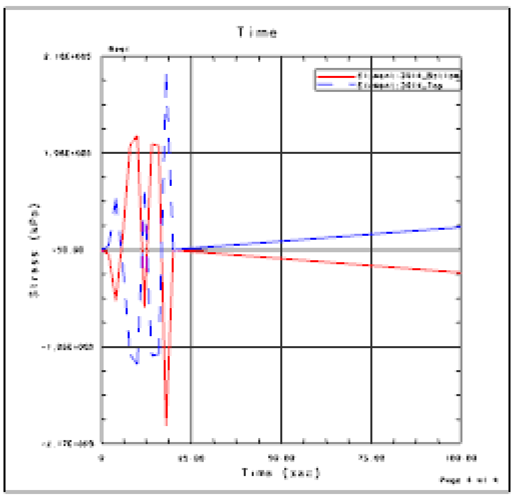
Use the new **Evaluate Fatigue Histories** command to export the elemental fatigue histories of the selected static or transient durability event to an external file. You file choices include:

- An AFU file. This file lets you visualize the history functions in the **XY Function Navigator**.

- An Excel spreadsheet. This file lets you manipulate the history functions using Excel functionality.

- A CSV file. This file lets you store the data in a comma-separated value ASCII file.

You can request stress or strain histories.



Stress history on top and bottom of an element

Why should I use it?

Exporting fatigue histories lets you examine the stress or strain histories that are used to perform fatigue computations. You can manipulate the generated fatigue load histories and compute damage using the **Evaluate Damage** command.

Where do I find it?

Application	Advanced Simulation

Simulation Navigator	Right-click a static or transient event node ® **Evaluate Fatigue Histories**

Result Path

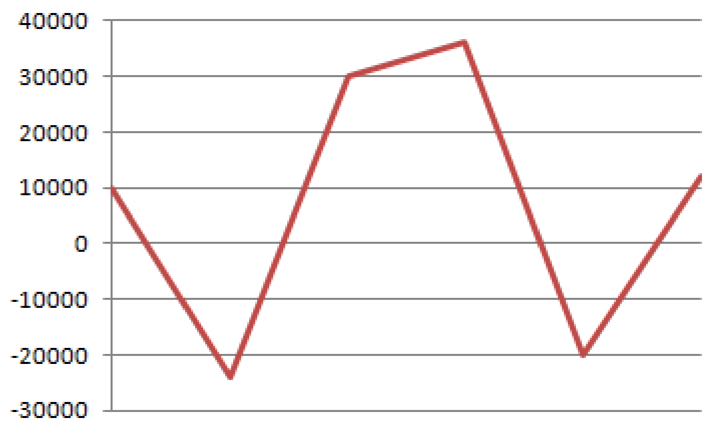
What is it?

Use the new **Result Path** excitation for a static durability event to create an excitation from existing solution steps.

In the **Result Path** dialog box, you select valid solution subcases. You can select each subcase multiple times and for each selection you can change the scale factor.

Example Assume that you have a static solution with the following two subcases:

- Subcase 1 has a stress loading of 10 000 Pa.
- Subcase 2 has a stress loading of 12 000 Pa.



To generate the result path excitation shown, you need to add the subcases and scale factors in the following order to the **Result Path** dialog box:

1. Subcase 1 with scale factor equal to 1.
2. Subcase 2 with scale factor equal to -2.
3. Subcase 1 with scale factor equal to 3.
4. Subcase 2 with scale factor equal to 3.
5. Subcase 1 with scale factor equal to -2.
6. Subcase 2 with scale factor equal to 1.

A given static event can have either load patterns or result path excitations, but not both.

Why should I use it?

This command enables quasi-static scaling of load cases to model spectrum loading.

Where do I find it?

Application	Advanced Simulation
Prerequisite	In the **Static Durability Event** dialog box, the **Excitation Type** list must be set to **Result Path**.
Simulation Navigator	Right-click a static event node ® **New Excitation** ® **Result Path** dialog box

Customer defaults for durability

What is it?

You can now define customer defaults for the NX Advanced Durability.

On the **General** tab, you can define the default temperature value that is used by a material with a temperature dependent property in NX Advanced Durability or the **Durability Wizard**.

On the other tabs, you can set the default values for options lists and boxes in a number of NX Advanced Durability dialog boxes.

Customer Defaults dialog box tab	NX Advanced Durability dialog box and default value location
Strength	**Static Durability Event** dialog box ® **Strength** tab **Transient Durability Event** dialog box ® **Strength** tab
Fatigue	**Static Durability Event** dialog box ® **Fatigue** tab **Transient Durability Event** dialog box ® **Fatigue** tab **Durability Damage Evaluation** dialog box ® **Fatigue Options** tab

Stress Axes	**Static Durability Event** dialog box ® **Stress Axes** tab **Transient Durability Event** dialog box ® **Stress Axes** tab **Durability Strain Gage Rosette Analyzer** dialog box ® **Options** tab ® **Axis Search** group
Strain Gage Analysis	**Durability Strain Gage Rosette Analyzer** dialog box ® **Options** tab ® **Output Requests** group

Where do I find it?

Application	Advanced Simulation
Menu	**File** ® **Utilities** ® **Customer Defaults**
Location in dialog box	**Simulation** ® **NX ADVANCED DURABILITY**

Durability comprehensive report

What is it?

Use the new **Report Durability Results** command to export selected durability results from the durability solution process to the following formats:

- An Excel spreadsheet using the **Export to Spreadsheet** option.
- A CSV file using the **Export to CSV File** option.

You can export the following durability results:

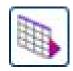
- Strength safety factor
- Margin of safety
- Fatigue safety factor
- Failure index
- Event damage
- Event life

In the previous release, you could export only event damage results using the **Report Durability Damage** command. The **Report Durability Results** command replaces the old **Report Durability Damage** command.

Why should I use it?

You can efficiently post process all durability result data that is computed by NX Advanced Durability.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Durability solution process with one or more events
Toolbar	**Durability** ® **Report Durability Results** 
Menu	**Insert** ® **Durability** ® **Solution** ® **Report Results**
Simulation Navigator	Right-click the Durability solution process node ® **Report Results**

NX Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal

Creating and modifying the parallel configuration file

What is it?

You can now create and modify a parallel configuration XML file for your run using the following new buttons:

- **Create New File**
- **Modify Existing File**

These two buttons replace the **Validate Machines** button that was available in the previous releases.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling[1] Advanced Thermal/Flow with ESC

1. DMP add-on license is required for parallel processing.

Solver	Analysis Type	Solution Type
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal[2]
		Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal[3]
		Advanced Axisymmetric Thermal
	Flow	Flow[4]
		Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow[5]
		Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Prerequisite	The **Run Solution in Parallel** check box must be selected on the **Solution Details** tab.
Toolbar	**Advanced Simulation** ® **Solve** ® **Edit Solution**
Simulation Navigator	Right-click a solution node ® **Edit**
Location in dialog box	**Solution Details** tab ® **Parallel Processing** group ® **Create New File** or **Modify Existing File**

CGNS export enhancements

What is it?

When you export a simulation to the CGNS format:

- Elements are now written in separate blocks mesh by mesh. In previous releases, all elements were written together in a single block.
- Meshes with transition elements are written in two separate blocks, one for each element type.
- Tapered beam meshes are also written in separate blocks, one block per beam.

2. DMP add-on license is required for parallel processing.
3. DMP add-on license is required for parallel processing.
4. DMP add-on license is required for parallel processing.
5. DMP add-on license is required for parallel processing.

- Boundary and embedded flow surfaces are written as the BCWall type. In previous releases, flow surfaces were not written to the *.cgns* file.

- Mesh and boundary condition names are now written to the *.cgns* file.

Why should I use it?

The CGNS format allows you to import your model into third party CFD software.

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
Menu	**File** ® **Export** ® **Simulation**
Location in dialog box	**File Type** list ® **CGNS**

Pressure results exported to CGNS

What is it?

When you export results from a simulation to the CGNS format, pressure results are now also written in addition to velocity and temperature results.

Why should I use it?

The CGNS format allows you to import your flow results into third party CFD software.

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 Solution**
Simulation Navigator	Right-click a solution node ® **Edit**
Location in dialog box	**Results Options** tab ® **Optional Output Format** group ® **CGNS** check box

Customer defaults for fluid domain mesh parameters

What is it?

You can now set the default values for the **Fluid Domain Mesh Parameters** options that are located on the **3D Flow** tab in the **Solution** dialog box.

Why should I use it?

You can customize the fluid domain mesh parameters 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 dialog box	**Simulation** ® **NX Thermal / Flow** or **NX ELECTRONIC SYSTEMS COOLING** ® **Solution** tab ® **3D Flow** group

Results options customer defaults location change

What is it?

A new **Results Options** tab is added. The customer defaults for the results options that used to be available on the **Solution** tab are now available on the **Results Options** tab.

Where do I find it?

Application	Advanced Simulation
Menu	**File** ® **Utilities** ® **Customer Defaults**
Location in dialog box	**Simulation** ® **NX Thermal / Flow** or **NX ELECTRONIC SYSTEMS COOLING** or **NX SPACE SYSTEMS THERMAL** ® **Results Options** tab

NX FE Model Updating

Rapid Create Design Variables

What is it?

Use the new **Rapid Create Design Variables** command to create multiple design variables for your Model Update solution process in one step.

In the **Design Variables Rapid Create** dialog box, you must select:

- The design variable type. To create **Design Variable — Property** modeling objects, you select the **Physical** option. To create **Design Variable — Material** modeling objects, you select the **Material** option.
- The initial value, lower bound, and upper bound. These values are assigned to all created design variables.
- One or more available physical properties or material cards.
- One or more property fields or material fields. The values of these fields are optimized.

To use the created design variables in your Model Update solution process, you must add them to your DESOPT 200 — Model Update solution. You do this on the **Bulk Data** tab in the **Solution** dialog box.

Why should I use it?

This command improves the FE Model Updating workflow for models that require you to create and manage hundreds of design variables.

Where do I find it?

Application	Advanced Simulation
Prerequisite	The DESOPT 200 — Model Update solution should be active.
Toolbar	**Model Update** ® **Rapid Create Design Variables**
Menu	**Tools** ® **Model Update** ® **Rapid Create Design Variables**

Set Value to 1.0

What is it?

Use the new **Set Value to 1.0** right-click command to set the initial value for all design variables back to 1.0 after a round of optimization.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the **Design Variables [#]** node ® **Set Value to 1.0**

Export design variables to spreadsheet enhancement

What is it?

When you use the **Export to Excel Spreadsheet** right-click command, more information on the design variables is now exported to the Excel spreadsheet. For each design variable, the exported information now includes:

- The name of the design variable
- The name of the physical property or material card
- The type information for the physical property or material card
- The field of the physical property or material type

In the previous releases, the information exported to the Excel spreadsheet for each design variable included: the status, ID, label, value, lower bound, upper bound, and weight.

Where do I find it?

Application	Advanced Simulation

Simulation Navigator	Right-click the **Design Variables [#]** node ® **Export to Excel Spreadsheet**

Update Design Variables from Spreadsheet

What is it?

Use the new **Update Design Variables from Spreadsheet** right-click command to import changes you made to the design variables in the Excel spreadsheet.

You can modify the following fields in the spreadsheet:

- **Status**. The acceptable values are: *Fixed* or *Free*.
- **Value**
- **Lower Bound**
- **Upper Bound**
- **Weight**

The last four fields must be real numbers with:

- Lower bound ≤ value ≤ upper bound
- Weight > 0.0

See Update design variable settings in an Excel spreadsheet for more information.

Why should I use it?

For large models, it is easier to manipulate the design variable data in an Excel spreadsheet. You can compute and update values, and make values track other fields. You can then import these changes to NX.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must first export the design variables to the Excel spreadsheet.
Simulation Navigator	Right-click the **Design Variables [#]** node ® **Update Design Variables from Spreadsheet**

Teamcenter Integration for Simulation

Manage AFEM Components

What is it?

Use the **Manage AFEM Components** command to synchronize CAE structures managed in Teamcenter with assembly FEMs in NX. This is similar to the synchronization currently provided for Teamcenter product structures and NX CAD assemblies.

When you open a new or modified CAE structure in NX, an **Information** window informs you that there are pending changes. Using the **Manage AFEM Components** command, you control which updates to include in your assembly FEM, and when to include them.

When you create an assembly FEM in NX, it is saved to Teamcenter as a CAE structure (that is, a CAEModel revision with a BOM view). In previous releases, however, changes to this CAE structure made in Teamcenter were not available to NX, and CAE structures created in Teamcenter could not be accessed in NX. This release of NX enables bi-directional interaction between Teamcenter CAE structures and NX assembly FEMs.

Beginning with this release, you can:

- Open CAE structures created entirely in Teamcenter (with or without datasets) as new assembly FEMs in NX. You are prompted to create an assembly FEM for the top-level CAEModel revision, and you can use the **Manage AFEM Components** dialog box to choose which CAEModel revision BOM lines to include as components in the assembly FEM.

- Add or remove CAEModel revisions to a CAE structure in Teamcenter, and update your NX assembly FEM to reflect pending changes using the **Manage AFEM Components** dialog box.

- Automatically import meshes managed as bulk data (for example, .nas files created by the batch mesher) as NX component FEMs. Bulk data files must be stored in CAEMesh datasets with a FEM_mesh reference type and an NX Simulations relation to the CAEModel revision.

- For non-associative assembly FEMs (assembly FEMs not mapped to a CAD assembly), you can synchronize component FEM positions with the transformation matrices stored in Teamcenter. This means that changes made to CAE structure component transformations are available to the NX assembly FEM.

  Note When you modify component transformation matrices in Teamcenter, you must save the modified BOM view revision before you can use them in NX.

NX setup

To configure NX to support this functionality:

1. Choose **File® Utilities® Customer Defaults**.

2. Navigate to **Simulation® General**.

3. On the **Environment** page, select **Enable Dialog Box to Manage Pending CAE Components**.

You must restart NX before these defaults take effect.

Teamcenter setup

To properly synchronize Teamcenter CAE structures and NX assembly FEMs, each CAEModel revision in the CAE structure must have a TC_CAE_Source relation to the corresponding CAD BOM line in the product structure.

- If you create component and assembly FEMs in NX, this relation is created automatically.

- If you create CAE structures in Teamcenter using the default data map, leaf nodes in the CAE structure have this relation, but assembly and subassembly nodes do not.

You can create the TC_CAE_Source relation manually in Teamcenter as needed, but it is recommended that your Teamcenter administrator modify the default *datamapping.xml* file to create this relation automatically for all CAE structures created using a data map or structure map.

Why should I use it?

In NX, component FEMs must be manually mapped to CAD component instances. Teamcenter enables you to automate this process using structure maps. For large structures, you can save significant time and resource by performing this mapping in Teamcenter.

Third-party or batch meshes stored as bulk data and managed in Teamcenter are immediately available for use in your NX assembly FEM.

In a collaborative engineering environment, colleagues, managers, and other stakeholders can quickly implement agreed-upon updates without using specialized software.

Where do I find it?

To update the NX assembly FEM to reflect changes made to the Teamcenter CAE structure, in the **Simulation Navigator**, right-click the top-level assembly FEM node and choose **Manage AFEM Components**. In the **Manage AFEM Components** dialog box, select the new or modified components to add or update and click **OK**. Click **Update Finite Element Model** if an update is required.

Application	Advanced Simulation
Prerequisites	Teamcenter Integration for NX. Teamcenter version 8.3, 9.1, and above. The customer default **Enable Dialog Box to Manage Pending CAE Components** must be selected. TC_CAE_Source relations must exist between assembly and subassembly nodes in the CAE structure and the corresponding CAD BOM line in Teamcenter. Meshes managed as bulk data in Teamcenter must be stored in CAEMesh datasets with a FEM_mesh reference type and an NX Simulation relation to the CAEModel revision.
Simulation Navigator	Right-click the top-level assembly FEM node and choose **Manage AFEM Components**

NX 8.5 Motion Simulation

Spring and Damper enhancements

What is it?

Note This topic is currently under construction.

Create spring and damper in a single step

You can now create a spring and damper motion object in a single step, using only the **Spring** command. The **Spring** dialog box now includes parameters for defining a damper object. When you create a spring and specify a damping coefficient in the **Spring** dialog box, NX creates two separate motion objects: a spring and a damper. Both objects are connected to the same action and base link or joint that you specify in the dialog box.

The ability to create a damper along with the spring is available only when you initially create the spring; it is not available when you edit an existing spring.

The existing **Damper** command remains available, but creates only a damper object.

Preload usability enhancement

The **Spring** dialog box has been redesigned to let you define preload directly. In previous releases, you defined preload indirectly, using the **Initial Length** option. Several new options are available:

- **Preload** — This option specifies the preload force on the spring. A negative value presses and a positive value pulls.

- **Length at Preload/Angle at Preload** — These options let you define the installed length or angle of the spring when it is under preload.

- **Free Length/Free Angle** — These display values report the length or angle of the spring at rest without any acting forces of tension or compression. When you specify a preload, the **Free Length** or **Free Angle** value updates automatically based on this formula:

  $$L_0 = \frac{F}{K} + L$$

  Where:

  L_0 = **Free Length**

  F = **Preload** at length L

  L = Length at preload F

  K = spring **Stiffness**

Where do I find it?

Application	Motion Simulation
Toolbar	**Motion**® **Spring**
Menu	**Insert**® **Connector**® **Spring**
Location in dialog box	**Spring** dialog box® **Damper** group® **Create Damper**

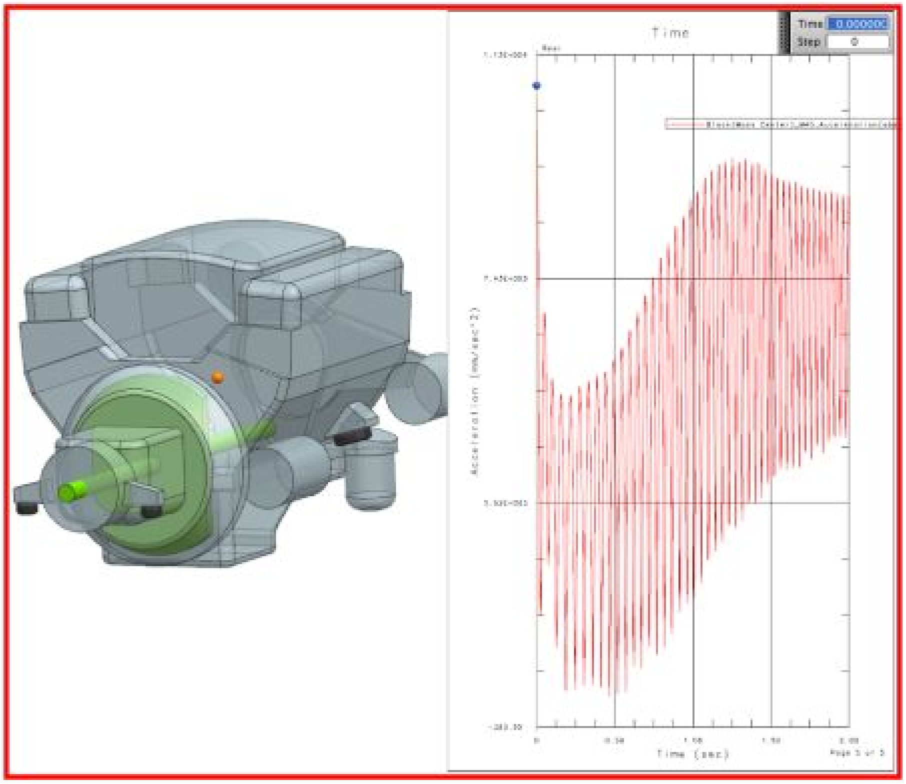
Graphing enhancements for Motion Simulation

What is it?

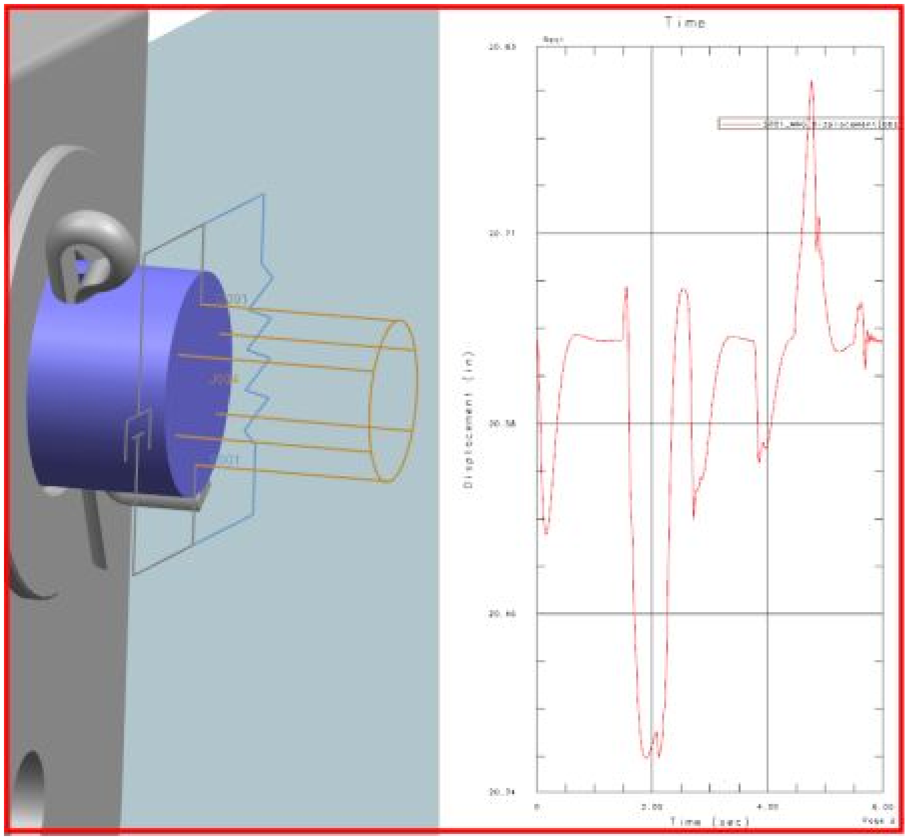
Note This topic is currently under construction.

This release includes two enhancements to the **Graphing** command.

- You can now select the center of mass of a link and then graph the displacement, velocity, and acceleration of the center of mass.



- You can now graph the rotational displacement of joints, springs, dampers, and bushings.



Where do I find it?

Application	Motion Simulation
Prerequisite	
Toolbar	
Menu	
Graphics window	
Location in dialog box	

Motion units enhancements

What is it?

Note This topic is currently under construction.

The standard NX **Design Logic** options (which include the units of measure selector, expression definition, and other options) have been added to input fields on the following dialog boxes:

- **Gear**

- **Rack and Pinion**
- **2-3 Joint Coupler**
- **Interference**
- **Motor**
- **Signal Chart**
- **Solution**

Where do I find it?

Application	Motion Simulation
Prerequisite	
Toolbar	
Menu	
Graphics window	
Location in dialog box	

Suppress/Unsuppress motion objects

What is it?

Note This topic is currently under construction.

You can now suppress individual motion objects and their dependent objects from being included in the active solution. By default, all motion objects, including links, joints, motion drivers, and so on, are now included in the active solution, unless you suppress them.

For example, you can suppress a link to remove it from the solution. That link's dependent joints and any other dependent objects, such as springs, markers, bushings, etc. will also be suppressed. If you then unsuppress one of the dependent objects, that object's parent object will also be unsuppressed.

Where do I find it?

Application	Motion Simulation
Motion Navigator	Right-click any motion object® **Suppress** (to remove the object from the active solution) Right-click any motion object® **Unsuppress** (to restore the object to the active solution)

Importing and exporting constraints and contact

What is it?

Note This topic is currently under construction.

You can now import or export mechanisms that contain the following motion object types:

- Point on Curve
- Point on Surface
- 2D Contact
- 3D Contact

In previous versions, the **Import Mechanism** and **Export Mechanism** commands did not support these types of objects.

Where do I find it?

Application	Motion Simulation
Motion Navigator	Right-click a simulation file **Export**® **Mechanism** Right-click a simulation file **Import**® **Mechanism**

Export integration with Process Simulate Kinematics

What is it?

Note This topic is currently under construction.

This release completes the integration between NX Motion Simulation and Tecnomatix Process Simulate Kinematics. You can now export your NX Motion Simulation model in PLMXML format for use within Process Simulate Kinematics. The new command is available only when you are running NX in managed mode.

The Tecnomatix Process Simulate on Teamcenter application allows you to design, analyze, simulate, and optimize manufacturing processes from the factory level down to lines and workcells. Part of the Process Simulate suite is a kinematics function that lets you create links and joints to model robots and other tools in the context of a manufacturing process. NX 7.5 added the ability to import a Process Simulate Kinematics model into NX Motion Simulation, to leverage the more powerful kinematics modeling capabilities of NX.

Where do I find it?

Application	Motion Simulation
Prerequisite	Teamcenter Integration for NX
Motion Navigator	Right-click a simulation file **Export**® **Process Simulate Kinematics**

Chapter

5 Teamcenter Integration for NX

Open NX Relations Browser from Teamcenter Navigator

What is it?

You can open the **NX Relations Browser** from the **Teamcenter Naviagator** to view the interpart links and expression relationships between parts without loading the parts in NX.

You can open the **NX Relations Browser** from an Item, Item Revision, or dataset. When the **NX Relations Browser** is initially opened, the view is set to **Show Relations of Currently Selected Parts**. This shows the selected part along with one level of children and one level of parents.

The **NX Relations Browser** lets you:

- Filter relations based on type and status
- Search parts that contain relations by part name
- Interactively transverse dependency paths
- Graphically show relation types and the number of each type of relation
- Automatically arrange nodes to minimize overlap.

By default, the **NX Relations Browser** only shows the parts that have interpart references to the selected part, not the details of each link. To see the details for individual wave links/interpart expresssions, use the product interface publishing functionality in NX. Once the links are published to Teamcenter (by saving the part and defined links and expressions), the links are shown in the browser even if the parts are not loaded.

For additional information on publishing product links, see the *Teamcenter Integration for NX, Publishing product interfaces and relations* online help.

For additional information on the **NX Relations Browser**, see the *Assemblies, Interpart modeling* online help.

Why should I use it?

You can use the **NX Relations Browser** to view interpart relationships without having to load the parts into NX.

Where do I find it?

Application	Teamcenter Integration
Teamcenter Navigator	Right-click an Item, Item Revision, or dataset ® **Relations Browser**

Rescue Session Data provides ability to save data when Teamcenter connection is lost

What is it?

The **Rescue Session Data** command lets you save your NX work when you do not have a connection to Teamcenter. The **Rescue Session Data** command replaces the **Save Outside Teamcenter** command. The new command provides the same basic function as the **Save Outside Teamcenter** command but is improved to enable you to more easily save and recover NX data.

The command is used when you lose your connection to Teamcenter and are not able to save NX data to the Teamcenter database, such as when Teamcenter crashes or hangs, or the connection to the database is lost. You can save your NX work to the operating system and re-import it back into Teamcenter.

Note The command is located at **File**® **Utilities**® **Rescue Session Data**. The command is not available by default so you may have to add it. Use the **Customize** command from a toolbar to add it to the menu listing.

When you use the command and save the part to the operating system a **recovery_import.clone** log file is also saved to the same location. This file contains information about the part so you can easily import the part back into NX.

After the **Rescue Session Data** operation is complete, the connection to Teamcenter is terminated. To import the part back into the Teamcenter database, restart NX and use the **File**® **Utilities**® **Import Rescued Data** command.

You can only use this method to save parts that have been modified in the current session; parts that have not been modified cannot be saved. Parts that are modified but are also part family members are not saved as they are tied to a template part. If a modified part is not saved, then its parent is not saved.

You cannot use a part saved in this manner in a native NX session. This is considered salvaged data from the managed environment and can only be used if it is imported back into the Teamcenter database.

For information on the Import Rescued Data command, see Import Rescued Data imports data saved when Teamcenter connection is lost.

Why should I use it?

You can save data in an NX model that has not been saved to Teamcenter when the Teamcenter connection is lost.

Where do I find it?

Application	Teamcenter Integration
Menu	**File® Utilities® Rescue Session Data**

Import Rescued Data imports data saved when Teamcenter connection is lost

What is it?

The **Import Rescued Data** command lets you import into Teamcenter the NX data you saved with the **Rescue Session Data** command.

Note The command is located at **File® Utilities® Import Rescued Data**. The command is not available by default so you may have to add it. Use the **Customize** command from a toolbar to add it to the menu listing.

When you use the **Rescue Session Data** command due to a lost Teamcenter connection, you save a part that has been modified in the session along with a **recovery_import.clone** log file that contains information about the part. The part and log file are saved to the operating system.

To import the part back into the Teamcenter database, restart NX and establish the Teamcenter connection, then use the **Import Rescued Data** command to select the part and log file and import the part from the operating system.

Why should I use it?

You can easily import NX data into the Teamcenter database that is saved with the **Rescue Session Data** command.

Where do I find it?

Application	Teamcenter Integration
Menu	**File® Utilities® Import Rescued Data**

Chapter

6 *Inspection and validation*

Check-Mate

Check-Mate Enhancements

What is it?

New checkers have been added to the **Get Information**test category:

- New **Modeling** checkers
  - **Report owning groups of an object** - An object can be the member of multiple groups. This checker reports all group member objects in a part. All the groups which contain the group object are also reported.
  - **Report a delete face command without a heal option** - A solid body may be converted to a sheet body when a face in the solid body is deleted by using the **Delete Face** option without selecting the **Heal** check box. Generally, converting the solid body to a sheet body may not be what a user wants. This checker reports all of the **Delete Face** features which do not use the **Heal** check box.
  - **Report non-positive expression values in a Sketch** - This checker reports all the non-positive dimensions in a sketch. In some cases there may be an advantage to use a geometric constraint instead.
  - **Report sketches that can be made external or internal** - This checker reports internal sketches which can be made external and external sketches which can be made internal.
- New **PMI** checker
  - **Report annotation text property of PMI objects** - This checker reports the annotation style for PMI objects.
- New **Assemblies** checker
  - **Report layer setting of a component** - When a component is added to the assembly, its layer can be set to **Original** or **As Specified**. This checker reports the layer option that was used for each component in the part.

The following enhancements have been made:

- One checker is enhanced in the **Template® DFM** (Design for Manufacturing) test category:
  - o **Check Sheet Body Anomaly checker** - This enhancement enables the selecting of a vector from the graphics window as the pull line direction, instead of setting a line or datum axis on a specific layer.

- One function in **Expressions** is enhanced:
  - o **mqc_rexp_matchSub** - This enhancement adds a new input parameter, **case_sensitive**. When the parameter is set to **False**, it enables the function to support case insensitive matching.

Where do I find it?

Resource bar	**HD3D Tools → Check-Mate**

Override results in Teamcenter

What is it?

You can now request override approval on Check-Mate results that are saved in Teamcenter. A designated user can approve or reject the request inside the Teamcenter rich client.

You can create one override approval request for each result. You can also edit, review, or delete requests.

An override approval can set test results to either automatically pass or automatically fail.

Why should I use it?

Some companies impose restrictions on the design process when certain checks fail. Override approval lets the design process continue if the designer can provide sufficient justification that the problem can be fixed later.

Before NX 8.5, you could not override results that were saved in Teamcenter, only results that are saved in the part file. For more information about the UGII_CHECKMATE_OVERRIDE environment variable, see the Check-Mate Help.

Where do I find it?

Prerequisite	You must run NX with Teamcenter Integration, using Teamcenter 8.3 or a later version.
Resource bar	**HD3D Tools**® **Check-Mate**
Menu	**Analysis**® **Check-Mate**® **View Check-Mate Results**
Location in dialog box	**Results** group® right-click a result® **Override Request**

Skip overridden Check-Mate tests in Teamcenter

What is it?

When a checker has an approved override request, you can use the **Skip Checking if Part has Overridden Results** check box to specify whether that checker is tested or skipped whenever Check-Mate tests are run.

Skipping an overridden test has no effect on the final status result that is saved in Teamcenter. When an approved override request exists, the final status result of the test is either **Passed** or **Failed**, depending on the setting of the **Override to** option in the approved request.

Why should I use it?

You can get better performance if you skip Check-Mate tests whose results are overridden. This is useful when you are interested only in the final status result of those tests.

Where do I find it?

Prerequisite	You must run NX with Teamcenter Integration, using Teamcenter 8.3 or a later version.
Resource bar	**HD3D Tools**® **Check-Mate**® **Settings** group® **Set Up Tests** 
Menu	**Analysis**® **Check-Mate**® **Set Up Tests**
Location in dialog box	**Run Options** tab® **Skip Checking if Part has Overridden Results**

Issue Navigator results displayed in Check-Mate

Use the **Check-Mate Results Viewer** command to send a validation log file that was created for an issue in the **Issue Navigator** to the **Check-Mate** HD3D tool.

Where do I find it?

Application	Issue Navigator
Prerequisite	You need an issue whose attachments include a validation log file.
Issue Navigator	Right-click a validation log file node® **Open With® Check-Mate Results Viewer**

Check-Mate checkers and functions

What is it?

NX 8.0.1 extends Check-Mate validation capabilities with new checkers in the **Get Information** category.

New checkers	Description
File Structure • **Report Referenced External Spreadsheets in Expressions**	Reports parts that contain expressions with references to external Excel spreadsheets.
Product Template Studio • **Report Dynamic Connections** • **Report Static Connections** • **Report PTS Template Part** • **Report Unresolved Dynamic Connections**	**Report Dynamic Connections** and **Report Static Connections** report mapping dependencies between Product Template Studio components. There are two types of dependencies between PTS components: • *Dynamic* connections are connections created by the Wave General Relinker according to a name pattern. • *Static* connections are Wave links and interpart expressions that already exist in the assembly when it is imported into Product Template Studio. **Report PTS Template Part** reports whether the current work part is a PTS template part. **Report Unresolved Dynamic Connections** reports unresolved dynamic connections between product template components.

Why should I use it?

You can select PTS checkers to report if the part is a Product Template Part. If it is, then the PTS checker can report the mapping dependencies between the PTS components.

Where do I find it?

Toolbar	**Check-Mate** toolbar® **Set Up Tests** 
Menu	**Analysis**® **Check-Mate**® **Set Up Tests**
Resource Bar	**HD3D Tools**® **Check-Mate**® **Set Up Tests** 
Location in dialog box	**Set Up Tests** dialog box® **Tests** tab® **Categories** group® **Get Information**® **File Structure** and **Product Template Studio** categories.

CMM Inspection Programming

New inspection path sub-operations

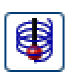
What is it?

Inspection paths include two new sub-operations.

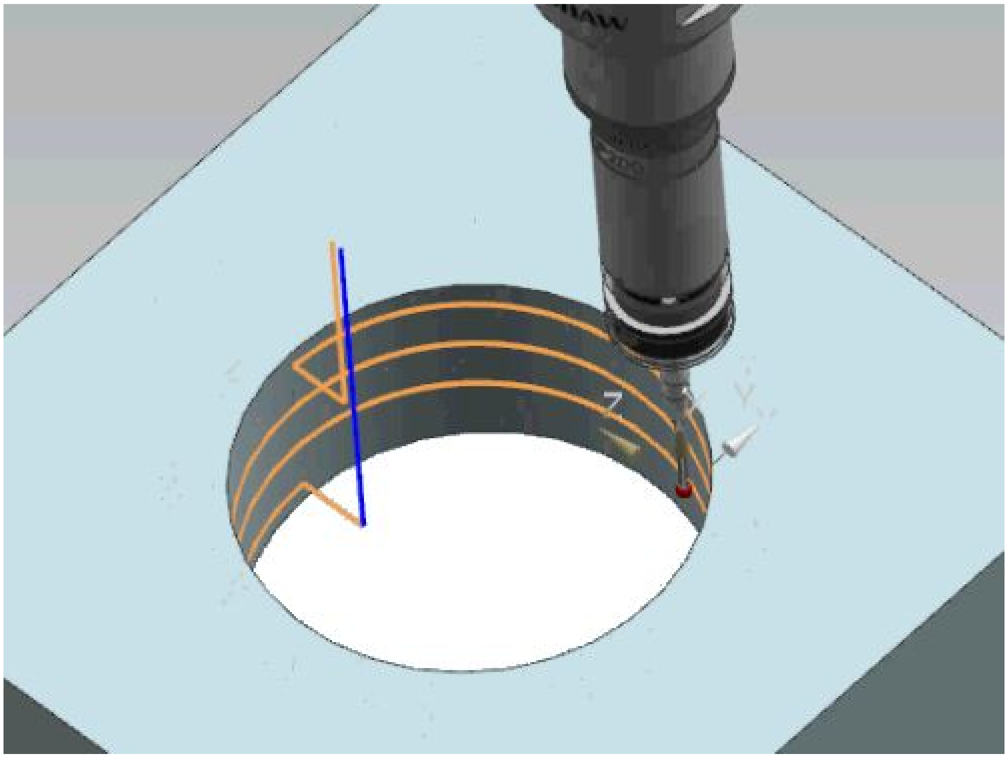
Use a **Linear 5-Axis Move to Point** sub-operation to move a five-axis probe to a specified location point while simultaneously rotating to specified A and B angles.

Note You can also define a five-axis linear move to point using the **Move Machine** command type in the **Insert Command** dialog box. Both result in the DMIS 5.2 output:

```
GOTO/CART,x,y,z,HEADCS,a,b
```



A **Scan Helix** sub-operation lets three-axis and five-axis probes approach any cylinder inspection feature to be measured, scan the inner or outer area, and exit using one continuous helical motion.



A typical helical scan on an inner cylinder inspection feature.

Why should I use it?

Helical scanning allows for greater accuracy at fast speeds than conventional touch-point measuring. Five-axis moves allow for quicker probe positioning than a three-axis move followed by a probe angle change.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must: • Create an inspection file and load an appropriate five-axis head and probe • For helical scans, create a cylinder inspection feature • For 5-axis moves, create an inspection path on a feature to be inspected after the move; for helical scans, create an inspection path on the cylinder feature • For 5-axis moves, in the appropriate order, add the move sub-operation and sub-operation for the

	feature to be inspected before or after the move; for helical scans, add the scan helix sub-operation
Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation** ® **Type** group® **Linear 5–Axis Move to Point** **Scan Helix**

5-axis scan sub-operation control point enhancement

What is it?

The **5 Axis Scan Curve** dialog box features a new section, **Control Points**, that lets you add multiple points and define tilt and advance angles for each point.

Why should I use it?

Control points help to ensure that probe angles vary smoothly at all points between control points.

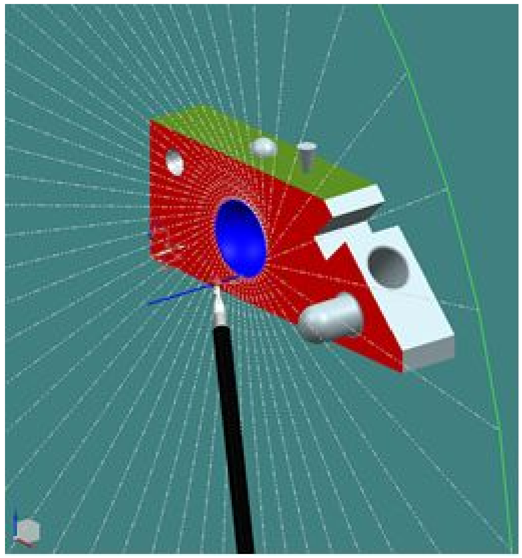
Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must: • Create an inspection file and load a 5–axis scanning head and probe • Create an arc, line, or curve feature • Create an inspection path on that feature • Add the sub-operation
Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation** ® **Type** group® **5 Axis Scan Curve**

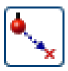
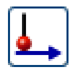
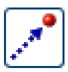
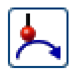
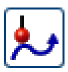
General scan sub-operation enhancements

Many enhancements have been added to refine scanning sub-operations.

- Points along scan curves, arcs, and lines are now visible to show the actual point density, and the orientation of the probe at each point is better represented to let you determine if a collision will occur.

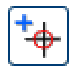


- Sub-operation icons have been updated. Move-to operations show dotted blue lines to indicate that they are not touching geometry, and scan operations show probes touching features.

	**Measured Point**		**Scan Line**
	**Linear Safe Move to Point**		**Scan Arc**
	**Scan Curve**		**5 Axis Scan Curve**

- In all scan sub-operation dialog boxes, a new **Curve Ending** list lets you select:
  - **Percentage**, and then type a **Start Percentage** and **End Percentage** where the probe should begin and end scanning.
  - **Distance from Start and End**, and then type a specific **Start Distance** and **End Distance**.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must: • Create an inspection file and load a scanning head and probe • Create an appropriate arc, line, curve, or cylinder feature • Create an inspection path on that feature • Add the appropriate sub-operation
Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation**  ® **Type** group® scan sub-operation

New CMM commands

The **Insert Command** dialog box includes two enhancements.

- A new command type, **Select Sensor**, now allows you to change probe tips and angles outside of an inspection by selecting the appropriate probe tool and a sensor that belongs to that tool. The sensor's tip and angle information appears in the dialog box to confirm that you have selected the appropriate sensor. If you postprocess your program in DMIS 5.2, the resulting code is output as in the following example:

  ```
  SNSLCT/SA(SENSOR_3)
  ```

  Note `SA` indicates a previously calibrated sensor. If you postprocess your program in DMIS 3.0, the result is `S`, an uncalibrated sensor.

- The **Move Machine** command type offers a new **5-axis Move** option for use with five-axis heads and probes. Like the **Linear Move** option, **5-axis Move** requires that you select a single point location to which the probe moves, but also lets you define an **A Angle** and a **B Angle** to which the probe rotates as it moves. This results in the DMIS 5.2 output:

  ```
  GOTO/CART,x,y,z,HEADCS,a,b
  ```

  Note You can also define a 5-axis move by creating a linear **5-axis Move to Point** sub-operation in the **Inspection Path** dialog box.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create an inspection file.
Toolbar	Insert® **CMM Command**
Menu	**Insert® CMM Command**
Inspection Navigator	Right-click any object and choose **Insert® CMM Command** to place the command in the **PROGRAM_HEADER** program group.

Collision Avoidance command

The collision avoidance command offers a variety of methods that you can apply to one or more inspection paths to avoid collision problems with geometry. When you specify an appropriate safe plane in combination with one or more avoidance methods, CMM Inspection Programming quickly calculates a solution that avoids most path collisions.

Where do I find it?

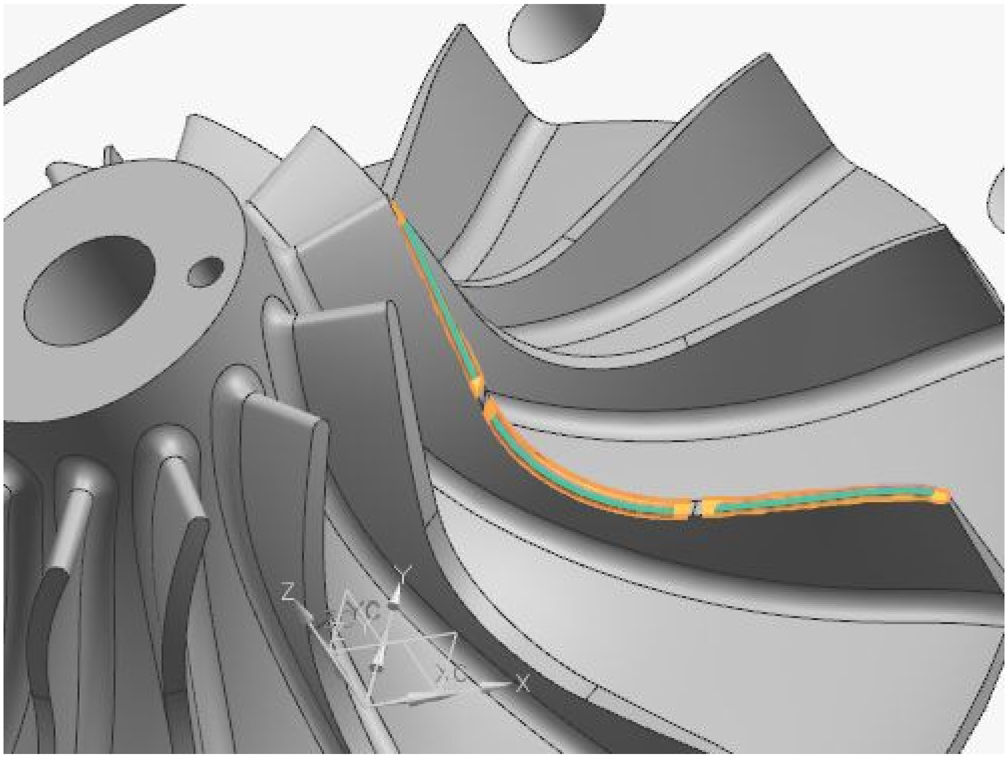
Application	CMM Inspection Programming
Prerequisite	In the **Inspection Navigator**, select one or more inspection paths
Toolbar	**Operations® Collision Avoidance**
Menu	**Tools® Inspection Navigator® Operation® Collision Avoidance**
Inspection Navigator	Right-click an inspection path and choose **Collision Avoidance**

Extract Feature operation

An *Extract Feature* operation extracts measurement data from curve inspection features that lie upon multiple part faces. This operation uses the Dimensional Measuring Interface Standard (DMIS) XTRACT statement.

You first scan one or more inspection curve features that span multiple part faces. You then use the resulting DMIS output on a CMM to extract measurement data to a single *design feature* on a single face. The data is extracted from only those sections of the curves that lie upon that face.

In the following example, a single curve spans three part faces. After the curve is scanned, three separate extract feature operations are defined. Each operation has its own surface design feature. The three design features, highlighted in gold in this example, are the features to which curve feature measurement data will be extracted. Each operation extracts data from only that area of the curve that lies upon that operation's design feature, less a defined start and end edge distance.

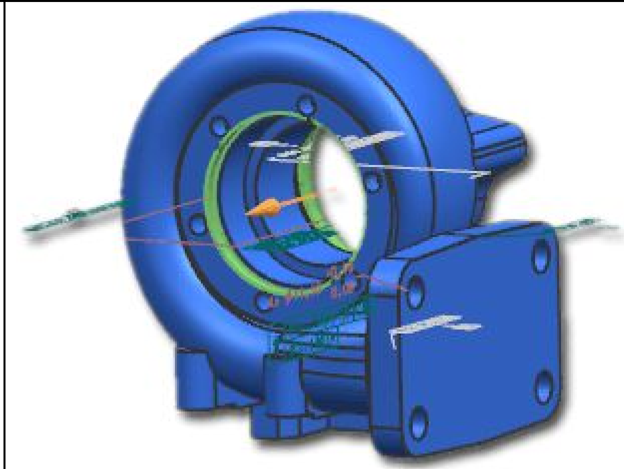
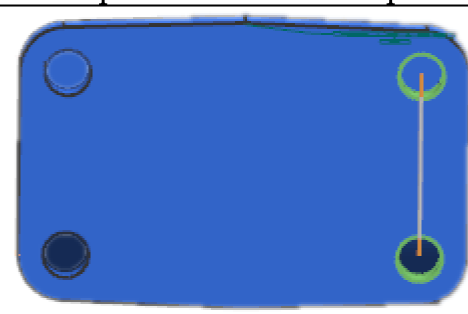


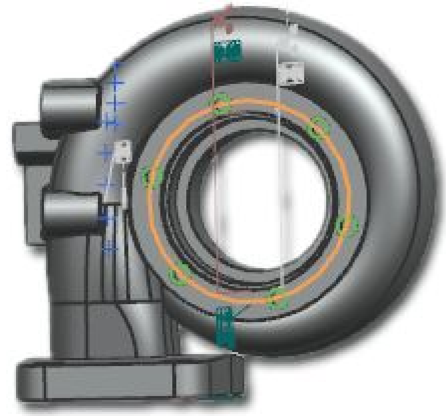
Where do I find it?

Application	CMM Inspection Programming
Prerequisite	You must create curve inspection features from which actual measurement data is derived, add inspection scan paths to those curves, and create a design feature on a face that the curve features enter or touch.
Toolbar	**Insert® Extract Feature** 
Menu	**Insert® Extract Feature**
Inspection Navigator	Right-click the design feature and choose **Insert® Extract Feature**.

Constructed feature enhancements

When creating a constructed plane feature in CMM Inspection Programming 8.0, you can in most cases create a new design feature rather than select an existing feature to serve as your constructed feature's nominal representation. In the **Constructed Feature** dialog box, the **Create New** option is now also available for the following constructed feature types and construction forms.

Constructed feature type	Construction form	Example
Point	All but **Transform**	 Design point feature created using the **Middle** construction form and two coaxial cylinder inspection feature midpoints
Line	All but **Transform**, **Offset**, and **Middle**	 Design line feature created using the **Perpendicular To** construction form, and a base and through cylinder feature

Circle	All but **Transform**	 Design circle feature created using the **Best Fit** construction form and the axis midpoints of six circle inspection features
Cylinder	All but **Transform**	 Design cylinder feature created using the **Best Fit** construction form and two sets of three constructed points. The first set lies on the closer cylinder inspection feature and the second lies on the farther coaxial cylinder inspection feature.

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 a node and choose **Insert**® **Constructed Feature**

Chapter

7 *Tooling Design*

Tooling shared functions

Standard Parts library

What is it?

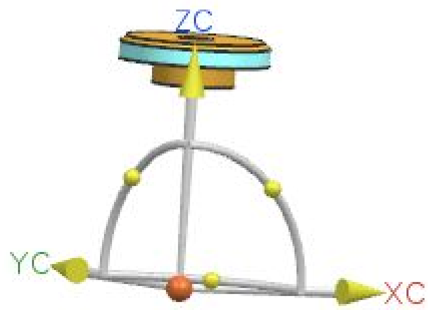
The **Standard Parts Library** for tooling wizards has the following usability enhancements.

With a standard part selected in the graphics window or in the **Assembly Navigator**, you can now edit it by pressing MB3, and choosing **Edit Tooling Component**.

The **Reuse Library** browser has the following improvements:

- When you select the top node in a library, pressing MB3 and selecting **Open Source Folder** opens the standard part data in native mode.
- When you select the top node in a library, pressing MB3 and choosing **Search Nodes Below Here** starts a search in the selected library.
- To locate the items that result from your search, you can select an item in the **Member Select** panel, press MB3 and choose **Locate**. The container for the item will be highlighted in the browser list.

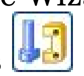
The standard part handle is by default positioned at 0,0,0 of the standard part. You can define a datum CSYS with the name MW_COMP_HANDLE to reposition the original point.



Why should I use it?

You can perform typical library tasks more easily and efficiently.

Where do I find it?

Application	Mold Wizard, Progressive Die Wizard, Electrode Design, Engineering Die Wizard
Toolbar	(Mold Wizard) **Standard Part Library**  (Progressive Die Wizard and Engineering Die Wizard) **Standard Parts** 
Menu	**Tools→Process Specific→Mold Wizard→Standard Part Library** **Tools→Process Specific→Progressive Die Wizard→Standard Parts** **Tools→Process Specific→Engineering Die Wizard→Standard Parts**
Resource Bar	**Reuse Library** 

Drawing Automation for Tooling

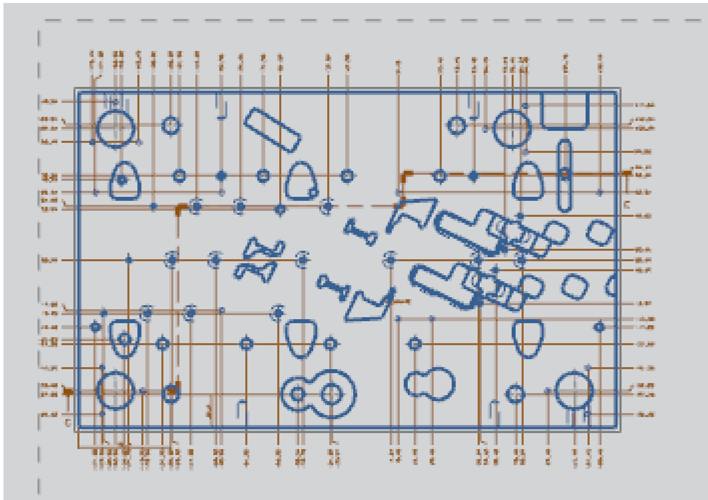
What is it?

Drawing functions in Mold Wizard and Progressive Die Wizard have several major enhancements:

- Performance of drawings, both assembly and component, has been improved.

- Workflows for **Assembly Drawing** and **Component Drawing** are supported in Native mode and in Team Center Engineering.

- Auto-dimensioning of circular holes and Wire EDM start holes is now available. Instead of having to manually select holes and dimension each one, you can choose **Select Holes Automatically** and all the holes in the drawing will have ordinate dimensions applied.



- **Hole Table** now supports multiple solid bodies in one part.

- The user interface for drawing functions has been streamlined and simplified.

Why should I use it?

Mold Wizard and Progressive Die Wizard can generate hundreds or even thousands of drawings. These enhancements make the process of creating drawings for tooling assemblies and components faster and more efficient.

Where do I find it?

Application	Mold Wizard, Progressive Die Wizard
Toolbar	(Mold Wizard) **Mold Drawing Drop-down** (Progressive Die Wizard) **Drawing Automation**
Menu	**Tools→Process Specific→Mold Wizard→Mold Drawing** **Tools→Process Specific→Progressive Die Wizard→Drawing Automation**

Mold Wizard

Cooling channels

What is it?

The following improvements have been made to the Mold Wizard cooling functions.

With the **Direct Channel**  command, you can:

- Add a tip angle or a rounded end to the channel you have created and see a preview of it.
- Extend the tip end of a channel that intersects the channel you have created and see a preview of it.
- Search for boundary bodies that intersect the channel you have created, and choose from four different options: in the same direction as the new channel, in the reverse direction, in both directions, or with no extension. You can see previews of these options.
- Have the option to add the newly created channel to the FALSE reference set in the **Direct Channel**, **Pattern Channel**, **Connect Channel** and **Define Channel** commands.

The **Adjust Channel**  command now provides a dynamic handle that lets you move the cooling channel and see a preview of it.

Why should I use it?

Creating and modifying cooling channels is easier and more intuitive.

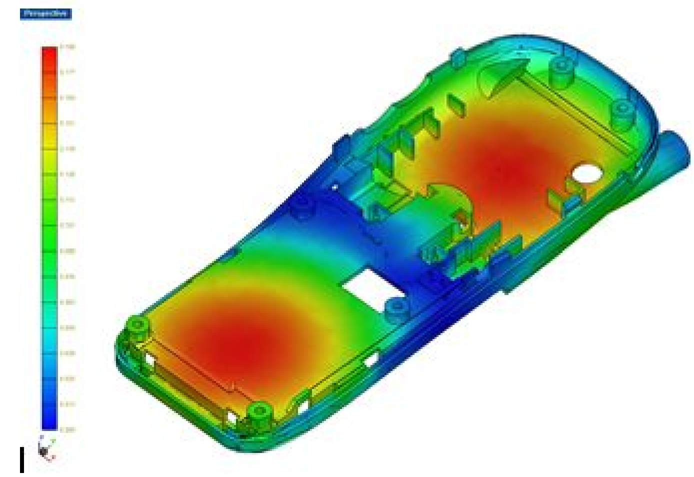
Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Cooling Tools**
Menu	**Tools→Process Specific→Mold Wizard→Cooling Tools**

Mold flow analysis

What is it?

Molded Part Validation now includes a new tool for analyzing mold flow and displaying results graphically.



Why should I use it?

Injection flow analysis functions are integrated into NX, and you can get the following flow types of flow analysis results:

- Melt Front Time
- Air Trap
- Welding Line
- Pressure Drop
- Melt Front Temperature
- Maximum Temperature
- Average Temperature

- Gate Contribution
- Frozen Layer Ratio
- Maximum Cooling Time

Where do I find it?

Toolbar	(Mold Wizard only) **Molded Part Validation** ® **Run Flow Analysis** and **Display Flow Analysis Results**
Menu	(Modeling, wizards, and other applications) **Analysis→Molded Part Validation→Easy Fill→Run Flow Analysis** and **Display Flow Analysis Results**

Mold Wizard workflow improvements

What is it?

A new method for creating edge patches (**Edge Patch**) lets you patch an opening by removing edges around it. This method is much simpler than the previous method, which created multiple features in the feature tree.

The **Split Face** command now has an option that allows you to find automatically the faces that need to be split and the edges to use to split these faces, instead of your having to search for them.

Why should I use it?

These improvements make the tools easier to use and produce better results.

Where do I find it?

Application	Mold Wizard
Toolbar	Mold Tools ® **Edge Patch** and **Split Face**
Menu	**Tools→Process Specific→Mold Wizard→Mold Tools→Edge Patch** and **Split Face**.

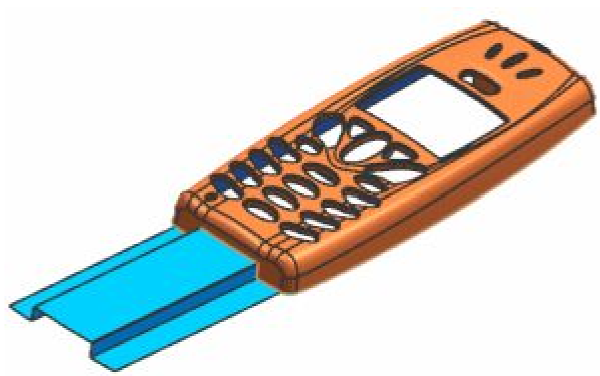
Design Parting Surface enhancements

What is it?

The process of creating parting surfaces has been improved and streamlined. You can now create most parting surfaces automatically. The **Design Parting Surface** command:

- Provides an **Auto Create Parting Surfaces** option that creates the parting surfaces automatically
- Selects a preferred surface type for a given parting segment
- Breaks up the parting loop into parting segments suitable for creating parting surfaces
- Lets you preview extruded and swept surface types
- Provides larger default sizes for the bounded plane and enlarged surface types
- Decides which side to keep when trimming parting sheets.

You can still break up the parting loop and create parting segments manually and override the automatic behavior.



Why should I use it?

You can create parting surfaces either automatically or manually, tailoring the command to your workflow and the complexity of the part.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Parting Tools** ® **Design Parting Surface**
Menu	**Tools→Process Specific→Mold Wizard→Parting Tools→Design Parting Surface**

Progressive Die Wizard

Sheet metal unforming

What is it?

New sheet metal unforming capabilities are available in Progressive Die Wizard and Engineering Die Wizard on the **Intermediate Stage Tools** toolbar (formerly **Sheet Metal Tools**). You no longer use the **Recognize Bends** option in **Direct Unfolding**. Instead, you select **Convert to Sheet Metal**, which now takes advantage of the new feature recognition and repair capabilities also newly available in the NX Sheet Metal application.

Additional improvements include the following. You can:

- Apply bend operations to parts with a zero bend radius or without planar stationary faces.
- Use the new **Universal Unform** command to assign linear bend or contour bend attributes to regions that cannot be unbent, and then successfully use **Unbend** to unform them.
- Unbend bend regions that include punch or stiffening type features such as beads, gussets, and ribs.
- Use the **Curve along curve** constraint in **One Step Formability Analysis** to help produce a better flattened freeform part.

Why should I use it?

When creating intermediate stages, you can successfully unbend more complex bends areas.

Where do I find it?

Application	Progressive Die Wizard, Engineering Die Wizard
Toolbar	**Intermediate Stage Tools**
Menu	**Tools→Process Specific→Progressive Die Wizard→Intermediate Stage Tools** **Tools→Process Specific→Engineering Die Wizard→Intermediate Stage Tools**

Electrode Design

Electrode Fixture

What is it?

You can now edit a fixture by selecting it in the graphics window or in the **Assembly Navigator**, pressing MB3, and choosing **Edit Electrode Component**.

Why should I use it?

You can do editing tasks more easily and efficiently.

Where do I find it?

Application	Electrode Design
Toolbar	**Electrode Fixture**
Menu	**Tools→Process Specific→Electrode Design→Electrode Fixture**

Design blank

What is it?

A new Manipulate Orientation handle is provided in the **Design Blank** command, to allow you to control the position and rotation of the blank.

Why should I use it?

You can easily move and reorient a blank in the X and Y directions.

Where do I find it?

Application	Electrode Design
Toolbar	**Design Blank** 
Menu	**Tools→Process Specific→Electrode Design→Design Blank**

Copy Electrode

What is it?

The **Mirror** type in the **Copy Electrode**  command has been enhanced. Now when you mirror copy an electrode head, the same blank found on the original will be installed on the copy.

Why should I use it?

The feature lets you easily create a mirrored blank and lets you check the interference between the mirrored electrode and the workpiece.

Where do I find it?

Application	Electrode Design
Toolbar	**Copy Electrode** 
Menu	**Tools→Process Specific→Electrode Design→Copy Electrode**

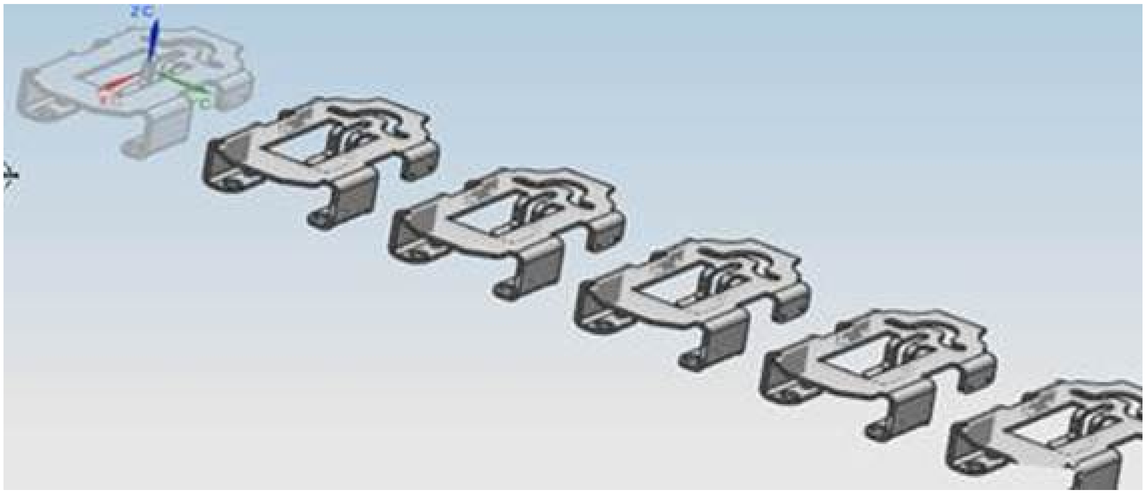
Engineering Die Wizard

Engineering Design workflow

What is it?

New functions are available in Engineering Die Wizard on the **Intermediate Stage Tools** toolbar (formerly **Sheet Metal Tools**).

- **Define Intermediate Stage**  lets you create associative intermediate unfolding stages in your assembly. You can see a preview of the stage layout, and link to a sheet body if you wish.



The command supports a default item type in Teamcenter for newly-created intermediate parts. You can establish a default naming rule in **Customer Defaults**.

- Use the streamlined **Direct Unfolding**  command with an imported part, or a part created with **NX Generic Tools**, to recognize and convert straight bends to sheet metal features. It allows you to convert non-parameterized parts into sheet metal parts that can be flattened.

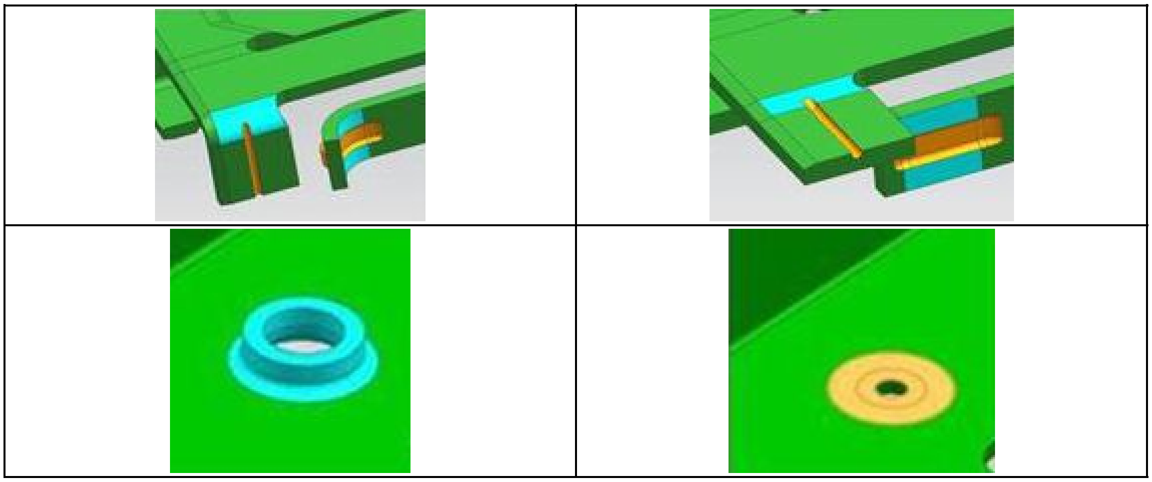
  Enhancements include the following:

  - The intermediate stage and feature recognition steps have been removed, simplifying the process.
  - You can now convert parts with zero bend radius.
  - You can now convert parts that have bends without a planar base face

- Use the new **Universal Unform**  command to assign linear bend or contour bend attributes to regions that cannot be unbent, and then successfully use **Unbend**  to unform them.

  You can convert parts that have the following characteristics:

  - Bend regions include deformation features
  - Bend regions include ribs, gussets, beads, or burring



The following commands available in Die Engineering are now also available Engineering Die Wizard. The functions have been streamlined, as appropriate, for Engineering Die Wizard.

- **Quick Binder Wrap**  lets you create a binder wrap that is planar, cylindrical, or conical, based on the shape of the sheet body. A *binder wrap* is the shape of the blank when the die closes and before the punch moves to give the sheet metal the desired shape. That shape should be developable, and this command makes it developable. A composite developable sheet body can be flattened without splitting or wrinkling.

- **Addendum Section**  automatically generates section curves tangent to the product face and perpendicular to the product boundary. You can create four types of addendum sections (Basic, Simple, Blend, and Extension) that you can use later to create an addendum surface. The output of this command is a single Addendum Section feature that is composed of one or more spline curves representing the addendum sections.

- **Addendum Surface**  lets you create an addendum surface using sections that you created with the **Addendum Section** command, as input. Two types of surface are available: **Sectional Sweep** and **Through Curve Mesh**.

- **Develop Trim Line**  lets you develop flange faces onto addendum faces.

  During the stamping process, a flat blank sheet body is formed into the desired shape, excess material is trimmed away, and then flanges are bent into position. It is during the trim operation—when flanges are laid out on the addendum—that an accurate trim line must be determined.

- **Draw Bead**  lets you apply a bead shape to a sheet body. You can also create two sheets representing the face of the upper and lower die containing the male and female bead shapes.

  A bead is a protrusion or indentation in the metal that resembles a tube.

  In automotive body engineering, beads are often applied to sheet metal parts to strengthen the material. During sheet metal forming, beads directly influence the draw by controlling the flow of material in the die.

A new **Remove Hole**  command finds holes or local deforms in a selected body or face automatically, and lets you delete them.

Why should I use it?

The addition of all these capabilities to Engineering Die Wizard automates and streamlines many functions and makes your workflow easier and more efficient. All the functions you need are available in one place.

Where do I find it?

Application	Engineering Die Wizard
Toolbar	**Intermediate Stage Tools** 
Menu	**Tools→Process Specific→Engineering Die Wizard→Intermediate Stage Tools**

Weld Assistant

Create User Exits

What is it?

A User Exit calls a user-created custom program and allows information to pass between the custom program and the NX weld features, when the NX user clicks **OK** or **Apply** to create specific features. You can create a User Exit for the following weld commands:

- **Fillet**
- **Groove**
- **Fill**
- **Bead**

- **Weld Point**
- **User Defined Weld**

Why should I use it?

Many companies standardize the attributes assigned to weld features. You can use customized User Exit programs to control these attribute values as soon as they are created.

You can invoke a custom program to enrich the information being added to the weld feature at the time of creation. Use Exit programs provide a means for supporting company-specific naming of weld features or standard company attribute assignments.

Where do I find it?

The custom program that gets called by the User Exit must be placed in the directory *ugweld\samples*.

Datum Locator APIs

What is it?

You can use the **Datum Locator** command to create a **Datum Pin**, a **Datum Surface**, and custom datums. These datum features are used for fixture and clamping tool design.

You can now create these datum features using NX Open APIs.

Why should I use it?

Use these APIs to programmatically create datum pin and datum surface features.

Where do I find it?

The callback location is *ugweld\samples*.

Datum Surface Locator, Datum Pin Locator

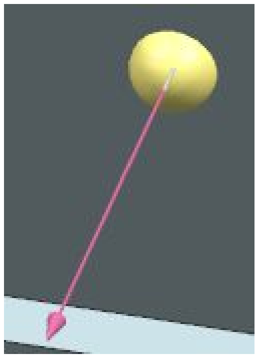
What is it?

Use these commands to create datum surface and datum pin features when you define the Body in White surface or axis datums.

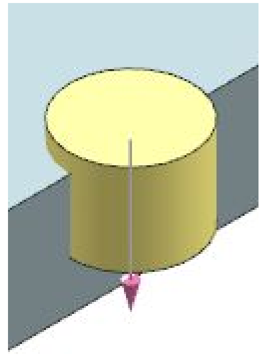
You can store and display:

- The input datum plane.
- The locating point.

- The direction vector.
- The datum locator definition and representation.



Datum Surface feature



Datum Pin feature

Why should I use it?

These datum features are used for fixture and clamping tool design.

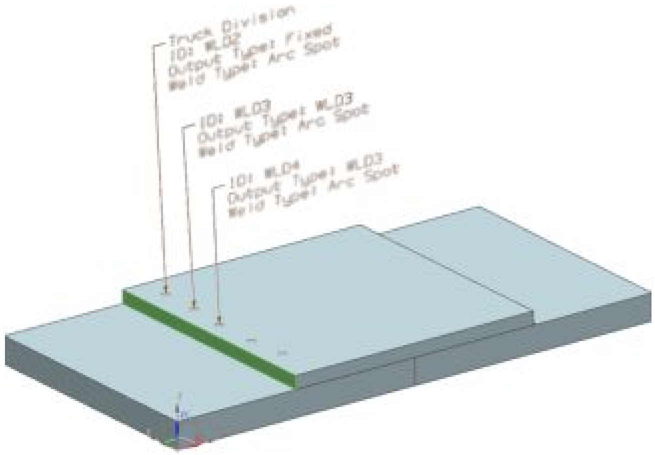
Where do I find it?

Application	Modeling
Toolbar	**BIW Locator® Datum Surface Locator** or **Datum Pin Locator**
Menu	**Insert® Welding® Datum Surface Locator** or **Datum Pin Locator**

Label in Weld Assistant

What is it?

Use the **Label** command to apply labels and annotations on welds, datums, and measurement points.



The labels are saved as PMI notes. PMI notes for welds, datums, and measurements are grouped in the **Part Navigator** separately as **Labels**.

Why should I use it?

You can create a high-level visualization of the weld and datum data in the CAD assembly or model. You can label multiple features at once, with information such as attributes and parameters, for example volume, connected parts, and IDs.

Where do I find it?

Application	Modeling
Toolbar	**Weld Assistant® Label**
Menu	**Insert® Welding® Label**

Structure Welding

Edit Joint Definition

What is it?

Use the **Edit Joint Definition** command to edit the parameters of multiple welding joints simultaneously.

Why should I use it?

The **Edit Joint Definition** command lets you specify the parameters that are used to create the geometry for beveling the ends of solid bodies to prepare them for welding.

Where do I find it?

Application	Modeling
Toolbar	**Structure Welding® Edit Joint Definition**
Menu	**Insert→Structure Welding® Edit Joint Definition**

Export Welding Joints

What is it?

Use this command to export weld joint data to an XML file. You can:

- Save the specified attributes and their order as a template for reuse.
- Specify the coordinate system that the output is in relation to.
- Control the number of points being output for a curve using a chordal deviation specification value.
- Configure the output information and save the XML file locally or in Teamcenter as an imported dataset.

- Change the attribute names when they are output to the XML file using the **Export Template Separator String** customer default.

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

Why should I use it?

XML is a neutral file format, and the weld joint data from such a file can be used as manufacturing input for robot controllers.

Where do I find it?

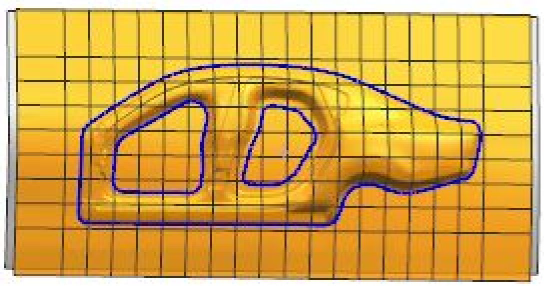
Application	Modeling
Toolbar	**Structure Welding® Export Welding Joints**
Menu	**Insert® Structure Welding® Export Welding Joints**

Die Design

Draw Punch enhancements

What is it?

When you use the **Draw Punch** command, you can now select multiple loops to create internal punch holes and internal closed ribs.



Why should I use it?

You can create draw punch with internal punch holes and closed ribs in side doors, windows, and other components of automobiles.

Where do I find it?

Application	Die Design
Toolbar	**Die Design® Draw Punch** 
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Draw Punch**

Vehicle Manufacturing Automation applications

What is it?

You must now choose the following applications from the **Start** menu.

- Die Engineering
- Die Design
- Die Validation

The toolbars and menus for these applications are no longer available from the Modeling application.

If you choose Die Design, reusable objects and **Standard Parts** are available in the **Reuse Library**.

Where do I find it?

Menu	**Start® All Applications® Vehicle Manufacturing Automation®** Die Engineering or Die Design or Die Validation

Die Engineering

Blank Nesting

What is it?

Use the **Blank Nesting** command to optimize the orientation of blank nesting to get maximum material utilization based on the curve profile, sheet body, or solid body of a part.

The **Blank Nesting** command provides six layout methods: **Rectangle**, **Parallelogram**, **Trapezoid**, **OneUp**, **TwoUp**, **TwoPair**

The **Blank Nesting** command lists the following values for maximum material utilization:

- Strip width
- Pitch between two blanks
- Blank area
- Maximum material utilization

Why should I use it?

When you maximize material utilization, you reduce the cost of a sheet metal part.

Where do I find it?

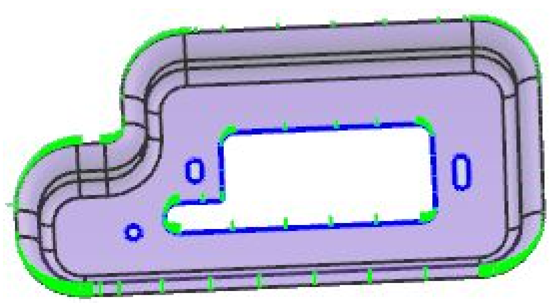
Application	Die Engineering
Toolbar	**Die Engineering® Design Assistant Tools® Blank Nesting** 
Menu	**Tools® Vehicle Manufacturing Automation® Die Engineering® Design Assistant Tools® Blank Nesting**

Trim Angle Check enhancements

What is it?

When you use the **Trim Angle Check** command, you can now:

- Select multiple close or open profiles as trimming curves.



- Use selection intent.
- Calculate the trim angle, and check trim angle information.
- View, hide, and edit trim angle check results in the **Part Navigator**.

The following customer defaults are now available:

- **Maximum Angle** specifies the upper limit of a valid trim angle.
- **Minimum Angle** specifies the lower limit of a valid trim angle.
- **Maximum Check Point Spacing** specifies the maximum distance between two check points.

Why should I use it?

You can use the **Trim Angle Check** command to analyze the angle at which scrap material is trimmed away from the part or addendum surface. You can check if a trim angle is compliant with manufacturing specifications by checking the color of the line segments.

Where do I find it?

Trim Angle Check command

Application	Die Engineering
Prerequisite	You must first create binder, addendum, and associated trim lines before you can use this command.
Toolbar	**Die Engineering**® **Checking Tools**® **Trim Angle Check** 
Menu	**Tools**® **Vehicle Manufacturing Automation**® **Die Engineering**® **Checking Tools**® **Trim Angle Check**

Customer defaults

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Customer Defaults** dialog box® **Die Engineering**® **Trim Angle Check**® **General** tab

Vehicle Manufacturing Automation applications

What is it?

You must now choose the following applications from the **Start** menu.

- Die Engineering
- Die Design
- Die Validation

The toolbars and menus for these applications are no longer available from the Modeling application.

If you choose Die Design, reusable objects and **Standard Parts** are available in the **Reuse Library**.

Where do I find it?

Menu	**Start**® **All Applications**® **Vehicle Manufacturing Automation**® Die Engineering or Die Design or Die Validation

Die Validation

Vehicle Manufacturing Automation applications

What is it?

You must now choose the following applications from the **Start** menu.

- Die Engineering
- Die Design
- Die Validation

The toolbars and menus for these applications are no longer available from the Modeling application.

If you choose Die Design, reusable objects and **Standard Parts** are available in the **Reuse Library**.

Where do I find it?

Menu	**Start**® **All Applications**® **Vehicle Manufacturing Automation**® Die Engineering or Die Design or Die Validation

Chapter

8 *Data translation*

DXF/DWG Export wizard

What is it?

You can use the new **DXF/DWG Export Wizard** to export NX parts to DXF or DWG format.

You can use the following **Export As** options:

- **2D** uses 2D Exchange translator to flatten the NX file.
- **3D** uses DXFDWG translator to export drawing and model data.
- **CGM** uses DXFDWG translator and CGM to export drawings.

You can also use the wizard to define:

- **Font** mapping to define DXF or DWG text font and text width.
- **Line Styles** mapping to define DXF or DWG line type and scale factor.
- **Crosshatch** mapping to define DXF or DWG crosshatch and scale factor.

Why should I use it?

Use this wizard to flatten an NX file and export it to DXF or DWF format.

Where do I find it?

Menu	**File**® **Export**® **AutoCAD DWF/DWG** **File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Gateway**® **Translator**® **DXF/DWG** tab® **Export As** option

NX to JT

Export JT dialog box

What is it?

You can use the new **Export JT** dialog box to configure the export settings for NX to JT translator. The export setting options are organized under following three tabs:

- **Files**: Lets you manage the configuration file and broadly specify how the data should be included in the JT file.

- **Tessellation**: Lets you specify the tessellation settings for better control over the tessellation data translated to JT.

  The **Use NX Resolution** setting allows you to reuse NX visualization tessellation settings in JT for selected levels. Optionally you can choose to tessellate the objects based on supplied chordal, angular, and length parameters using the **Define Level** option.

- **NX Objects**: Lets you specify which NX objects to be translated and how the nodes should be structured in the JT file.

Why should I use it?

Manually editing the configuration file is tedious and error—prone process, instead use the **Export JT** dialog box to configure the export settings from the NX user interface.

Where do I find it?

Menu	**File® Export® JT**

JT Configuration enhancements

What is it?

The **JT Configurations** dialog box is updated for the following:

- The **Compression Control** setting is removed from the **Files** tab. The functionality which this setting was offering is already supported using the `JtFileFormat` option in the configuration file.

- The **Use NX Resolution** setting is added to the **Tessellation** tab which allows you to reuse NX visualization tessellation settings in JT for selected level.

Why should I use it?

Use the **Tessellation** settings in the updated **JT Configuration** dialog box to get similar tessellation quality in JT as that of NX, for the JT file generated during part save process using the **File® Save/SaveAs** option.

Where do I find it?

Menu	**Preferences® JT**

Writing all NX reference sets to a single JT file

What is it?

While translating NX files to JT, you can now translate all reference sets of NX part or assembly to a single JT file, when written to the file system.

While visualizing the JT file, you can use the reference set loading capabilities of visualization applications like Teamcenter Visualization to view the data of specific reference set.

Why should I use it?

Use this setting if you want single JT file that can hold all reference sets from the translated NX part or assembly and view objects present in specific reference set.

Where do I find it?

Recommended settings	Following settings are associated to this functionality and their recommended values in the *tessUG.config* file are given below:
	`mergeSheets=false`
	`mergeSolids=false`
	`useRefsets="ALL"`

Command line	`run_ugtopv.bat -config=tessUG.config -externalPDM NXPart.prt`

Writing NX reference sets as layer filters in JT

What is it?

While translating NX files to JT, you can now optionally translate reference sets from NX part or assembly as layer filters in JT.

The names of the layer filters translated to JT file are same as that of the corresponding reference sets in NX.

The `activateFilters` configuration control should be set to `"REFSET"` in the *tessUG.config* file.

Note To translate the NX layer categories as well as reference sets to JT file, set the `activateFilters` configuration control to `"LAYER,REFSET"` in the *tessUG.config* file.

Why should I use it?

Use this setting if you want to write reference sets in the NX part or assembly as layer filters in JT and make use of layer filtering capability of JT viewers like Teamcenter Visualization to view the objects from particular layer filter of interest.

Where do I find it?

Recommended settings	Following settings are associated to this functionality and their recommended values in the *tessUG.config* file are given below:
	`mergeSheets=false`
	`mergeSolids=false`
	`useRefsets="ALL"`

Command line	`run_ugtopv.bat -config=tessUG.config -externalPDM NXPart.prt`

Writing NX attributes as string data type in JT

What is it?

While translating NX files to JT, you can now translate any standard or user defined attributes in NX as 'string' data type. The precision of the data defined in NX is retained as it is when translated as string data type.

Attributes defined in NX	Attributes in JT: Default data type	Attributes in JT: String data type
Surface_Area: 1.479213.568 mm^2	Surface_Area: 1.47921e+006	Surface_Area: 1.479213.568
Volume: 8854848.298 mm^3	Volume: 8.85485e+006	Volume: 8854848.298

The `getAttributes` configuration control should be set to "`true`" in the *tessUG.config* file.

Why should I use it?

Use this setting to preserve the accuracy of NX attribute data translated to JT.

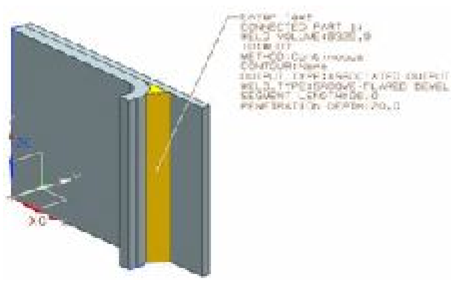
Where do I find it?

Command line	`run_ugtopv.bat -config=tessUG.config -externalPDM NXPart.prt`

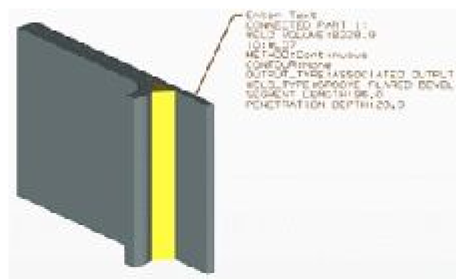
JT support to weld, datum, and measurement labels in NX

What is it?

While translating NX files to JT, you can now transfer the PMI label annotations created for welds, datum, and measurement features to JT file as note PMI. The JT file also retains the associative information between the PMI and the weld geometry.



Weld label definition in NX



Weld label translated as note PMI to JT

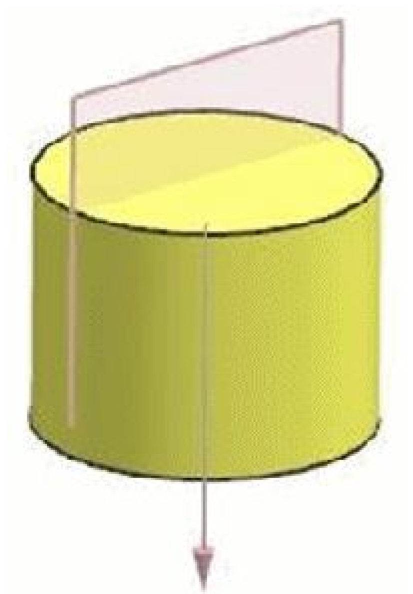
Where do I find it?

Application	Supported in all modes of JT generation.
Prerequisite	The `activateNotePMI` configuration control should be set to "`true`" in the *tessUG.config* file.

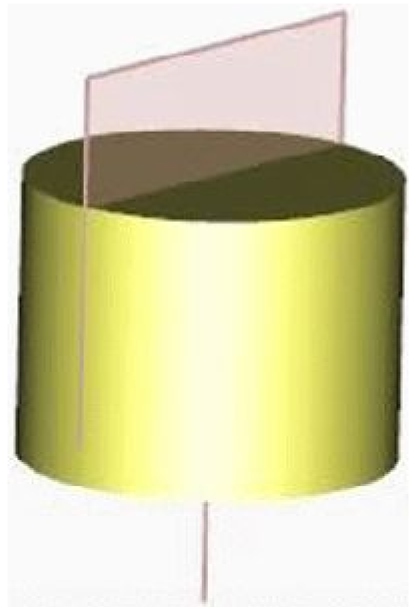
JT support to datum surface and pin locators in NX

What is it?

While translating NX files to JT, you can now translate the datum locators along with datum planes and the direction vector information in a part or an assembly to JT file.



Datum pin locator in NX



Datum pin locator translated to JT

Where do I find it?

Application	Supported in all modes of JT generation.
Prerequisite	The `activateCsysPMI` configuration control should be set to "`true`" in the *tessUG.config* file.

JT support to new NX line width

What is it?

While translating NX files to JT, you can now translate all new line widths applied to wireframe geometry, datum, and PMI objects.

NX Line width (mm)	JT line width (mm)
0.13	0.13
0.18	0.18
0.25	0.25
0.35	0.35
0.50	0.50
0.70	0.70
1.00	1.00
1.40	1.40
2.00	2.00

Where do I find it?

Application	Supported in all modes of JT generation.

Reference Set Definition customer default

What is it?

When you import JT files to NX, you can now control the method that is used to assign geometric data in JT files to NX assembly reference sets. You can set the new **Reference Set Definition** customer default to one of the following options.

- **Use Multi-CAD Reference Sets** uses multi-CAD attributes to assign objects in the JT file to NX reference sets. This is the default selection.

- **Use Layer Filter Reference Sets** uses JT layer filters to assign objects in the JT file to NX reference sets. Use this default only if your JT files contain layer filters.

Where do I find it?

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Gateway® JT Files® Import** tab

PMI and Reference Geometry customer default

What is it?

When you translate JT files to NX, you can now control the loading of PMI and reference geometry data. You can set the new **PMI and Reference Geometry** customer default to one of the following options. This default works only if the **Import Assembly Structure from JT Files** customer default is not selected.

- **Always Load** loads the PMI and reference geometry data whenever the JT file is loaded, even if you load the JT file using lightweight data.

- **Never Load** prevents the loading of PMI and reference geometry data in the current NX session. If you later decide that you want the data, you must deselect this default and translate the JT file again.

- **Only Load with Exact Data** delays the loading of PMI and reference geometry information until you load the exact B-Rep data.

Why should I use it?

When you import JT files into NX using lightweight representation, significant time is spent on processing PMI and reference geometry data. To reduce processing time, you can delay the loading of this data until you load the exact data.

Where do I find it?

Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Gateway**® **JT Files**® **Extract Exact Data** tab

Chapter

9 Mechatronics Concept Designer

Mechatronics Concept Designer objects in Reuse Library

What is it?

You can insert reusable Mechatronics Concept Designer objects from the reuse library to your model. Mechatronics Concept Designer objects stored in the reuse library contain the following in addition to the 3D geometry data:

- Physics data such as rigid, and collision body types
- Sensors and actuator information with their interfaces
- Cam and function information that are performed by a functional unit
- Operation that are performed by a functional unit

Since all Mechatronics Concept Designer related data is stored in the NX part file, creation of reusable Mechatronics Concept Designer components is the same as any other CAD component. When a reusable component is inserted to a model from the Reuse Library, the properties mentioned above are automatically added to the model. User can still modify any of these properties in the model.

Why should I use it?

Reuse Library provides the ability to create reusable knowledge-enabled components and reuse them in current or future designs.

Where do I find it?

Application	Mechatronics Concept Designer
Resource bar	Reuse Library

Runtime Parameters

Use the **Runtime Parameters** command to create a reusable physics object that is a container of signal parameters. After you create a **Runtime Parameters** object in your model, the signal parameters can be referenced by any physics objects in your model.

If you change the value of one or more parameters of the **Runtime Parameters** object in your model, an instance of the object is created at the parent assembly level of the part that contains the object. This instance is called a **Runtime Parameters Override** object. The updated parameter values only appear in the override object.

Where do I find it?

Application	Mechatronics Concept Designer
Toolbar	**Mechatronics Concept Designer** ® **Runtime Parameters**

Proxy Object

Use the **Proxy Object** command to create reusable higher level physics objects such as motors or actuators. A proxy object contains both geometry and runtime properties.

When you add a proxy object to your model, you can change the runtime parameter values of it in your model. If you change the runtime parameter values of the proxy object in your model, an instance of the object is created at the parent assembly level of the part that contains the proxy object. This instance is called a **Proxy Override** object. The updated parameter values only appear in the override object.

Furthermore, a proxy override object can have a rigid body attachment. In such case, the proxy object geometry is merged with the rigid body during runtime.

Where do I find it?

Application	Mechatronics Concept Designer
Toolbar	**Mechatronics Concept Designer** ® **Proxy Object**

Replacement Assistant

What is it?

When you use the **Replacement Assistant** command, you can now:

- Transfer operations from a concept model to a detailed model.
- Import physics objects that exist in the replacement component to your model.
- Automatically recreate references to higher level assembly objects when you replace a concept component that has references to its parent assembly with a detailed component. In your model, any physics object that is referenced in the concept model from the assembly level higher than the one you are replacing will be recognized by the replacement assistant. References will automatically be mapped to from the appropriate replacement component to the higher level assembly.

Where do I find it?

Application	Mechatronics Concept Designer
Prerequisite	Assemblies must be selected on the **Start** menu. You must use the **Replace Component** command to replace a concept model component with a detailed model component.
Toolbar	**Assemblies** ® **Replace Component** 
Menu	**Assemblies** ® **Components** ® **Replace Component**

Mechatronics Concept Designer ECAD Integration

What is it?

As the result of this enhancement, you can now:

- Use the **Export to ECAD** command to export sensor and actuator information from Mechatronics Concept Designer to ECAD software. This command generates an XML that must be imported to EPLAN software. This file can then be authored in EPLAN if necessary and imported to ECAD.
- Import sensor and actuator information from EPLAN to Mechatronics Concept Designer to create new sensors or actuators or to modify existing ones.

In addition, a new library is added to the reuse library that is called **ECAD Reuse Library**. This library contains component geometry. When a sensor or an actuator is imported from ECAD, user can find a component in the reuse library and use it as the 3D model for the imported sensor or actuator.

Where do I find it?

Application	Mechatronics Concept Designer
Toolbar	**Mechatronics Concept Designer ® Export to ECAD** **Mechatronics Concept Designer ® Import from ECAD**

Function Navigator enhancements

What is it?

As the result of this enhancement, you can now:

- Access and modify functional models in the **System Navigator**. Functional models can be created or configured in NX or Teamcenter and are accessible from the **System Navigator**. The following objects from the functional model are visible in NX:
  - Function Item
  - Function structure
- View logical models in the **System Navigator**. The following objects from the logical model are visible in NX:
  - Logic Item
  - Logic structure
- View physical models in the **System Navigator**.
- Access or create trace links on base objects. Trace links can be accessed from function or logic item revisions.

Where do I find it?

Application	Mechatronics Concept Designer
Prerequisite	You must be in Teamcenter Integration mode.
Graphics window	**System Navigator**

OPC server connection with Mechatronics Concept Designer

You can connect physics objects to external signals through an Open Process Control (OPC) server. An OPC server connection lets you apply values continuously from external signals of a control system to physics objects in the mechatronics model. This connection also lets you export some physics runtime property values to the OPC server. These property values are used by control systems as instructions or feedback.

To create the OPC server connection, you must:

1. Define the OPC server. You must set the server type, ID, and host name.
2. Set up the OPC client and select the OPC tags to use.
3. Connect physics objects to OPC tags.

OPC Client Parameters

Use the **OPC Client Parameters** preferences to setup the OPC client and select the OPC tags you want to connect to physics objects. The available OPC tags can be from the following sources:

- OPC server
- OPC file
- User defined

Where do I find it?

Application	Mechatronics Concept Designer
Prerequisite	You must define the OPC Server settings in the **Customer Defaults**.
Menu	**Preferences® OPC Client Parameters**

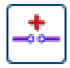
External Connection

Use the **External Connection** dialog box to define a connection between physics objects and external signals. You can connect OPC tags to physics objects that have float, integer or Boolean parameters. Such objects include:

- Rigid, collision, and trigger bodies
- Transport surfaces
- Mechatronics joints and constraints

- Object sources and object sinks

Where do I find it?

Application	Mechatronics Concept Designer
Toolbar	**Mechatronics Concept Designer® External Connection** 
Menu	**Insert® External Connection**

Chapter

10 Programming Tools

Specify arguments used in NX Open programs and journals

What is it?

When you run an NX Open program or journal file you can specify the arguments used in it. This enables you to use a program/journal many times and customize it for use in a particular application by supplying a specific set of arguments.

This functionality is applicable when you:

- Run an NX Open program
- Play back a journal file
- Create an action for a new customized command
- Run an NX Open program or journal file using NX Open Menuscript

The arguments are read in as a string array and argument counter, parsed, tracked, and passed into the NX Open program/journal when it is executed.

To specify arguments when:

- Running an NX Open program:
  1. Choose **File**® **Execute**® **NX Open**.
  2. Click **Enter User Function Arguments**.
  3. Enter your arguments in the **User Function Arguments** box.
- Playing back a journal file:
  1. Choose **Tools**® **Journal**® **Play**.
  2. Enter your arguments in the **Enter Journal Arguments** box.
- Creating an action for a new customized command:
  1. Choose **Tools**® **Customize**.
  2. Select the **Commands** tab.

3. Select **New Button** at the bottom of the **Catagories:** list.

4. Click on **New User Command** in the **Commands:** list and drag it to a toolbar.

5. Right-click on the **User Command** button on the toolbar and choose **Edit Action**.

6. Enter your arguments in the **Enter Action Arguments** box.

- Running a program from NX Open Menuscript:

  Include the arguments in the entry for running the program. For example

```
BUTTON my_button
ACTIONS dotnet.vb("This is arg 1","dotnet_arg2")
        cpp_dll.dll("cpp_arg1") program.exe
```

Why should I use it?

You can specify a particular set of arguments when running an NX Open program or journal file.

Where do I find it?

Application	All
Menu	**File® Execute® NX Open**; **Tools® Journal® Play**; **Tools® Customize**
Location in dialog box	**User Function Arguments** box; **Enter Journal Arguments** box; **Enter Action Arguments** box

SNAP

What is it?

Simple NX Application Programming (SNAP) is improved to provide additional functionality when creating SNAP programs. The following functionality is added:

- Tangent lines
- Trimming and dividing curves
- Converting and joining curves
- Arclength calculation utilities

- Point sets
- Layer categories
- Enhancements to intersection calculations
- Enhancements to minimum distance calculations
- Functions for creating block-based dialog boxes programmatically. These functions allow you to created simple dialog boxes just by writing a few lines of code without using Block Styler.
- New global properties are added to control the tolerances used in constructing geometry (Snap.Globals.DistanceTolerance and Snap.Globals.AngleTolerance).
- SNAP Deviation Checking functions are enhanced and performance is significantly improved.
- The way that Snap.Number objects are converted to strings is changed. You can now always use a period (instead of a comma) as the decimal point, regardless of your local environment. This improves compatibility with other areas of NX.
- A function is added to clear the contents of the Info window without closing it.
- The SNAP Reference Guide includes documentation to support the new functionality, plus many new examples that illustrate the use of Snap.Geom objects.

In addition, you can now create and run simple SNAP programs from the Journal Editor without purchasing a SNAP author license. This allows you to explore and learn SNAP to get a feel for its simplicity and usability. The basic functionality is there to complete the exercises in the *Getting Started with SNAP* guide.

Why should I use it?

You can use the additional functionality in SNAP programs, create simple SNAP programs from the Journal Editor, and learn more about SNAP from the updated SNAP Reference Guide.

Where do I find it?

The *Getting Started with SNAP* guide is provided in PDF format. The SNAP Reference Guide is provided in Microsoft Help format (.chm).

Block UI Styler enhancements

Customizing visibility of command buttons

What is it?

You can now customize the visibility of command buttons **OK**, **Apply**, **Cancel**, and **Close** during dialog box design, by using the enumeration property Navigation Style.

The Navigation Style property has the following values:

- OK Apply Cancel (default value)
- OK Cancel
- Close

NX modifies code generator options for callbacks based on the selected Navigation Style value.

Where do I find it?

Application	**Block UI Styler**
Location in dialog box	**Dialog** dialog box® **Properties** group **General** list® **Other** list® **Navigation Style**

Static APIs for block property

What is it?

NX Block UI Styler now provides Static APIs to get and set block properties.

The Static APIs are available in the NXOpen.BlockStyler namespace in NXOpenUI.dll.

Use **Generate Block Specific Code** for block specific code generation during dialog box design time when you use the Static APIs. The default value for **Generate Block Specific Code** is True.

Example VB.net code for an integer block with BlockID as `integer0`:

```
Private integer0 As NXOpen.BlockStyler.IntegerBlock

integer0 = CType(theDialog.TopBlock.FindBlock("integer0"),NXOpen.BlockStyler.I
```

`integer0.Value` will return the Value property of this block.

To set the Value property of this block, the code should be one of the following:

```
integer0.Value = 500

Dim val as integer  = integer0.Value
```

Why should I use it?

Use Static APIs to prevent runtime failures due to incorrect use of property names.

Where do I find it?

Application	**Block UI Styler**
Location in dialog box	**Dialog** dialog box® **Code Generation** tab® **General** list® **Generate Block Specific Code**

Accessing block properties after dialog box closure

What is it?

You can now access properties of a block even after the dialog box is closed and the program control comes out of the `Show()` method.

Access the block properties by calling the `GetBlockProperties()` method and passing the Block ID to it. If the ‘Dialog’ is passed as Block ID, properties of the dialog box will be available even after the dialog box is closed.

Example

```
Public Shared Sub Main()
        Dim theMyDialog As MyDialog = Nothing
        Try
            theMyDialog = New MyDialog()
            ’ The following method shows the dialog immediately
            theMyDialog.Show()

       ’## Earlier Block’s properties were not accessible here ##
pList As BlockStyler.PropertyList  = theMyDialog.GetBlockProperties(“toggle0”)

        Catch ex As Exception
            ’---- Enter your exception handling code here -----
theUI.NXMessageBox.Show("Block Styler", NXMessageBox.DialogType.Error, ex.ToSt
        Finally
            If theMyDialog IsNot Nothing Then
                theMyDialog.Dispose()
                theMyDialog = Nothing
            End If
        End Try
    End Sub
```

New block properties

What is it?

NX Block UI Styler now provides additional properties.

- List Box
  - SelectedItemIndex
  - SelectedItemString
  - SelectedItemIndexes
  - SelectedItemStrings
  - SelectedItemBooleans
- Multi-Line String
  - ValuesConcatenated
- Integer Block
  - ReadOnlyValue
- Double Block
  - ReadOnlyValue
- Specify Axis Block
  - StepStatus

Three new static methods are added for the Select Object block. The VB signatures for these APIs are:

- AddFilter(By Val Filter As Integer)
- AddFilter(Type As Integer, SubType As Integer, SolidBodyType As Integer)
- ResetFilter()

Chapter

11 PCB Exchange

NX 8.5 PCB Exchange

IDF defined area types for placement regions and other outlines

What is it?

PCB Exchange now allows you to define area types for placement regions and other outlines when importing an IDF file.

In the IDF file, the name assigned to the placement regions or other outlines can match the area type that you define with the restriction area variable in the *pcbx_ug.ini* file. When you import the IDF file, PCB Exchange assigns the new area type and settings to the restriction area instead of the Placement or Other standard IDF area type and settings if the assigned names match between the two files.

Example The IDF file has two placement regions defined; their names are `COPPER_CLAD_1` and `COPPER_CLAD_2`. In the *pcbx_ug.ini* file, two restriction area variables are defined with area types `COPPER_CLAD_1` and `COPPER_CLAD_2`.

IDF file

```
.PLACE_REGION MCAD
TOP COPPER_CLAD_1
0 46.0000 6.0000  0.0000
0 58.0000 6.0000  0.0000
0 58.0000 95.0000 0.0000
0 46.0000 95.0000 0.0000
0 46.0000 6.0000  0.0000
.END_PLACE_REGION

.PLACE_REGION MCAD
TOP COPPER_CLAD_2
0 63.0000 26.0000 0.0000
0 75.0000 26.0000 0.0000
0 75.0000 99.0000 0.0000
0 63.0000 99.0000 0.0000
0 63.0000 26.0000 0.0000
.END_PLACE_REGION
```

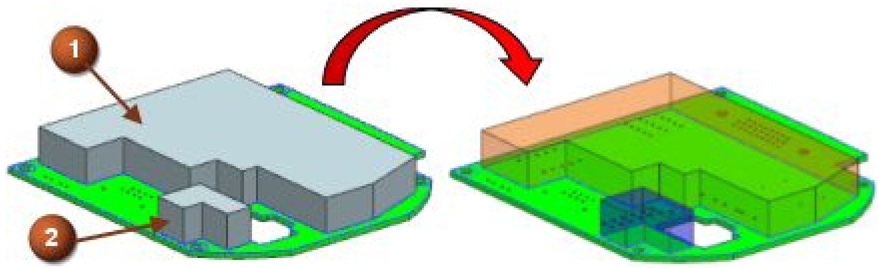
pcbx_ug.ini file

```
KeepInAreaType = COPPER_CLAD_1, place_region, 25, 114
KeepInAreaType = COPPER_CLAD_2, place_region, 24, 211
```

When you import the IDF file, PCB Exchange assigns the following to bodies (1) and (2):

- `COPPER_CLAD_1` as the name and area type, 25 as the layer, and deep orange as color 114 of the placement region named `COPPER_CLAD_1` in the IDF file.

- `COPPER_CLAD_2` as the name and area type, 24 as the layer, and blue as color 211 of the placement region named `COPPER_CLAD_2` in the IDF file.



Why should I use it?

Use IDF defined area types to define additional area types for placement regions and other outlines when importing an IDF file.

Where do I find it?

Application	PCB Exchange
Toolbar	**PCB Exchange** toolbar® **Import ECAD Model**
Menu	**PCB Exchange** ® **Import ECAD Model**

The *pcbx_ug.ini* file can be found in the following locations:

- NX installation directory

- Location specified by MAYA_PCB_DIR

- Network location specified by MAYA_PCB_ENV_DIR

Automatic attribute assignment by layer

What is it?

You can now create sketches or solid bodies, place them on separate NX layers, and then use the new **Automatic Area Creation** command to automatically assign restriction area attributes to the sketches or bodies, according to their NX layers.

The following attributes are assigned:

- Area type
- Color
- Ownership
- Transparency
- Height

Use the `KeepInAreaType`, `KeepOutAreaType`, `OtherAreaType` variables, in the *pcbx_ug.ini* file, to control the layer and color that is associated to the restriction area type.

The **Automatic Area Creation** command assigns attributes either to the bodies or sketches on the layers. You control whether it is bodies or sketches with the `NxWriteAreasAs` and `NxWriteAreaAttributesOnTheSketch` variables in the *pbcx_ug_model.ini* file.

Example The following table shows the variables defined in their respective file.

File name	Variables defined
pcbx_ug.ini	`KeepOutAreaType = Routing, route_keepout, 25, 211` `DefaultKeepOutOwner = MCAD`

pcbx_ug_model.ini	`NxWriteAreasAs = Bodies`
	`NxWriteAreaAttributesOnTheSketch = No`
	`NxWriteTransKeepout = 70`

After you use the **Automatic Area Creation** command, the following occurs to all bodies on NX layer 25:

- They are tagged as keep-out restriction areas of type, `Routing`.
- They have their color changed to blue, 211.
- They are owned by MCAD.
- They have their transparency changed to 70%.
- They are assigned a height obtained from the body in the direction of the board normal.

In previous releases, you could only manually define restriction areas and their attributes with the **Keep-in Area Attributes**, **Keep-out Area Attributes**, and **Other Area Attributes** commands.

Why should I use it?

Use the **Automatic Area Creation** command to efficiently assign restriction area attributes to multiple sketches or bodies on specified NX layers.

Where do I find it?

Application	PCB Exchange
Toolbar	**PCB Exchange® Tools® Automatic Area Creation**

The *pcbx_ug.ini* and *pcbx_ug_model.ini* files can be found in the following locations:

- NX installation directory
- Location specified by MAYA_PCB_DIR
- Network location specified by MAYA_PCB_ENV_DIR

Colors and layers for restriction areas

What is it?

You can now control the NX layer and NX color for restriction areas when importing an IDF file, of versions 2.0 and 3.0, when you set the new settings `NX Layer #` and `NX Color #` for the following restriction area variables in the *pcbx_ug.ini*.

```
KeepInAreaType = Area Type, ECAD Layer, NX Layer #, NX Color #
KeepOutAreaType = Area Type, ECAD Layer, NX Layer #, NX Color #
OtherAreaType = Area Type, ECAD Layer, NX Layer #, NX Color #
```

`NX Layer #` Layer number between 1 and 256. You can leave it blank, in which case PCB Exchange uses the default layer specified in the *pcbx_ug_model.ini* file.

`NX Color #` Color number between 1 and 216. You can leave it blank, in which case PCB Exchange uses the default color specified in the *pcbx_ug_model.ini* file.

When importing an IDF file, for a given area type, the sketch or body, which represents the restriction area, is placed on the specified NX layer and assigned the specified NX color.

Why should I use it?

Use the `NX Layer #` and `NX Color #` settings in the *pcbx_ug.ini* file to better control NX layer and colors for restriction areas when importing an IDF file.

Where do I find it?

The *pcbx_ug.ini* and *pcbx_ug_model.ini* files can be found in the following locations:

- NX installation directory
- Location specified by MAYA_PCB_DIR
- Network location specified by MAYA_PCB_ENV_DIR

Support for splines in board profiles

What is it?

When you export to an IDF file, PCB Exchange now automatically approximates spline profiles in the board definition using arc curves, since IDF does not support splines.

In previous releases, PCB Exchange approximated spline profiles using lines.

You control the accuracy of the approximation with the `NxWriteSplineToArcsTolerance` variable in the *pcbx_ug_model.ini* file. The tolerance corresponds to the maximum distance allowed between two arcs and the original spline profile, in model units. A small tolerance increases the number of arcs used in the approximation and increases the time required to export and reimport.

Where do I find it?

Application	PCB Exchange
Prerequisite	A board with attributes and the PCB Exchange application active.
Toolbar	**PCB Exchange ® Export ECAD Model**
Menu	**PCB Exchange® Export ECAD Model**

The *pcbx_ug_model.ini* file can be found in the following locations:

- NX installation directory
- Location specified by MAYA_PCB_DIR
- Network location specified by MAYA_PCB_ENV_DIR

NX 8.0.1 PCB Exchange

GenCAD import enhancement

What is it?

When you import a GenCAD model, you can now import non-closed traces as solid bodies.

In the previous release, non-closed traces were imported only as curves. If you specified that they were imported as solid bodies, NX issued a warning in the import log file.

NX now tries to combine trace segments into one solid body. It uses the width and thickness of the trace segments to create the solid body. The width is specified in the GenCAD model. You can control the thickness of the trace solid bodies by modifying the value of the `NXWriteDefaultTraceThk` variable in the *pcbx_ug_model.ini* file.

If NX fails to create one solid body, it creates individual bodies for each segment.

Where do I find it?

Application	PCB Exchange
Prerequisite	GenCAD model
Menu	**PCB Exchange** ® **Settings**
Location in dialog box	**Other Entities** tab ® **Traces as** list ® **Bodies**

The *pcbx_ug_model.ini* file can be found in the following locations:

- NX installation directory
- Location specified by MAYA_PCB_DIR
- Network location specified by MAYA_PCB_ENV_DIR

Importing ECAD models from Teamcenter

What is it?

If you use Teamcenter Integration, you can now import ECAD models directly from Teamcenter.

In the **Import ECAD Model** dialog box, you must enter the ECAD model designator name with the complete path from Teamcenter.

Note You cannot browse the Teamcenter database structure in this dialog box.

Where do I find it?

Application	PCB Exchange
Prerequisite	Teamcenter Integration for NX OAn empty NX part file, with the PCB Exchange application active
Toolbar	**PCB Exchange** ® **Import ECAD Model** 
Menu	**PCB Exchange** ® **Import ECAD Model**
Location in dialog box	**ECAD Model** group ® **ECAD Data From** list ® **Teamcenter**