What's New in NX 7

Publication Number
whatsnew7 0

Chapter 2: Introduction to What's New

The What's New Guide briefly summarizes the new features in each release.

This guide highlights what each function does, why it should be used, and where it can be found in the user interface. This guide also conveys the benefit of each new capability.

New features are also introduced in NX Maintenance Releases between major releases. To review subsequent enhancements that were introduced after the initial release of NX 6, click on the entry in the Table of Contents titled **Enhancements in NX 6.0.x Maintenance Releases**. You can also access the What's New Guide for each maintenance release from the What's New Guide launch page.

Chapter 3: HD3D

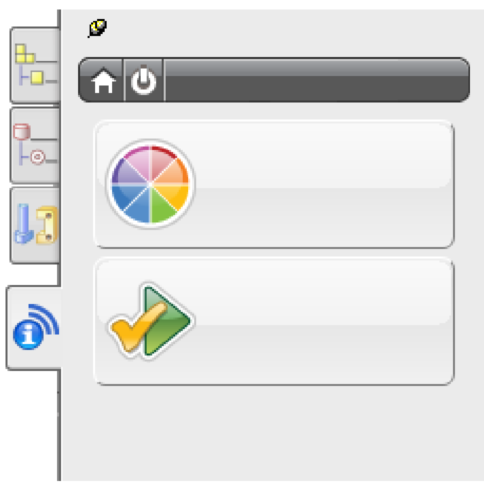
HD3D overview

What is it?

Use the HD3D tools in NX to display and interact with information directly on the 3D model.

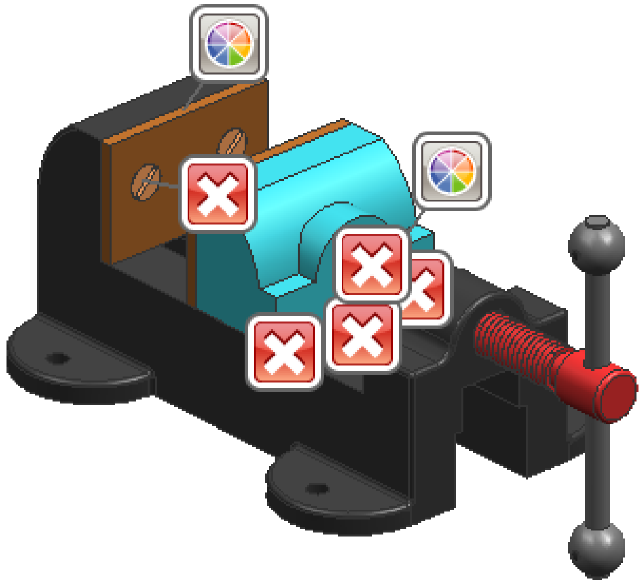
You can:

- Access the HD3D tools from the HD3D Tool Manager on the Resource bar. Currently, **Visual Reporting** and **Check-Mate** HD3D tools are available.

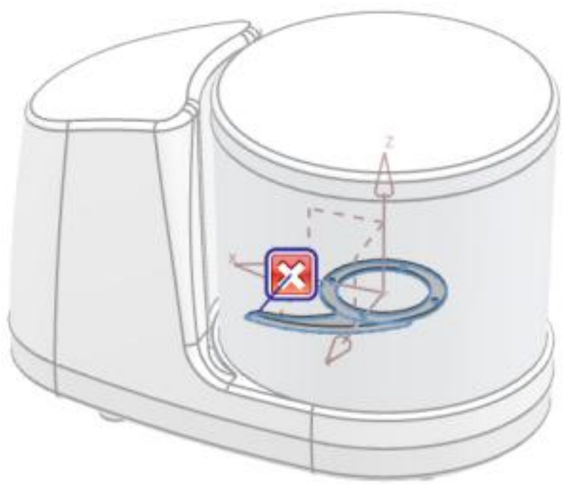


- View and interact with tags in the graphics window.

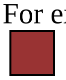
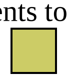
  Tags identify objects for which an HD3D tool has information. Check-Mate tags identify areas with problems or check result information. Visual Reporting tags appear as color wheels.

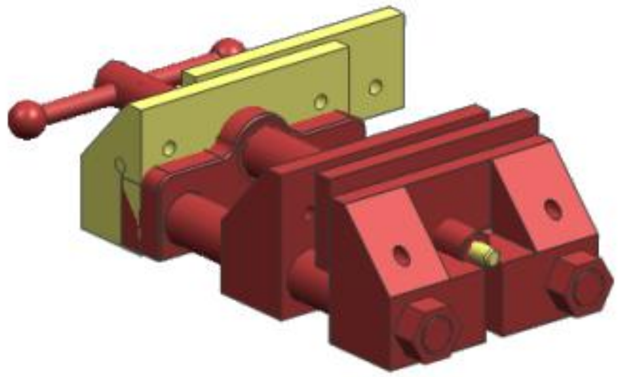


- Control how you view tags and other information inside the 3D model using the **See-Thru** display styles.



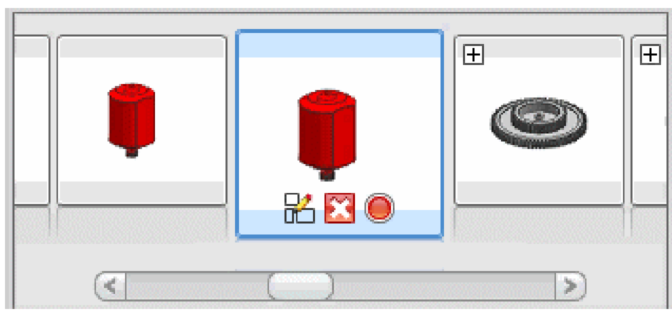
- Use visual reporting to get a visual report of information on the model.

  For example, you can color code components to display their load state. In the following graphic,  components are partially loaded and  components are fully loaded.



- Display information in different styles in the HD3D dialog box.

  For example, to scroll through thumbnails to find the items you want, use the **Flow List+Tree** view style in the **Check-Mate** tool.

  When you first activate an HD3D tool, the view styles are shown in the tool's dialog box. You can also display the tool's results in its own separate window.

- Set preferences for the HD3D tools using the **HD3D Tools** dialog box.

Why should I use it?

You can use the HD3D tools to interact visually with information presented in the graphics window that may previously have been presented in dialog box lists and information reports.

Where do I find it?

HD3D Tool Manager

Resource bar	**HD3D Tools** tab 

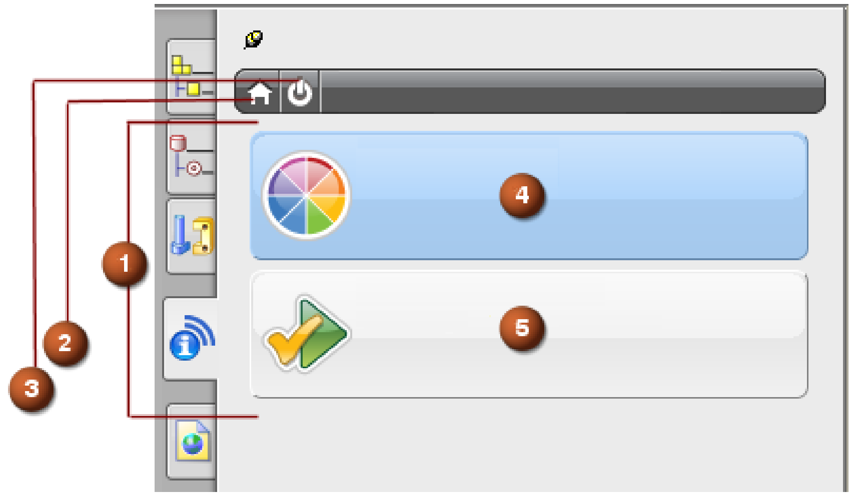
See-Thru option

Toolbar	**View→See-Thru** 

HD3D Tool Manager

What is it?

The *HD3D Tool Manager* provides access to HD3D tools. The **Visual Reporting** tool helps you visualize product information. The **Check-Mate** tool helps you validate product information.



#	Component
1	HD3D Tool Manager
2	**Return to HD3D Tool Manager** button
3	**Deactivate** button
4	**Visual Reporting** tool
5	**Check-Mate** tool

From the HD3D Tool Manager, you can:

- View the status of each tool.
- Activate or deactivate a tool.

  When you activate an HD3D tool, the tool's dialog box replaces the HD3D Tool Manager.
- Refresh a report after you make changes to your model.
- Deactivate an HD3D tool so that no report information appears in the tool's button.
- Access the **HD3D Tool Properties** dialog box where you can select the applications in which you want to access the HD3D tool.

Where do I find it?

Resource bar	**HD3D Tools** tab

HD3D view styles

What is it?

View styles let you view and navigate the information made available by an HD3D tool.

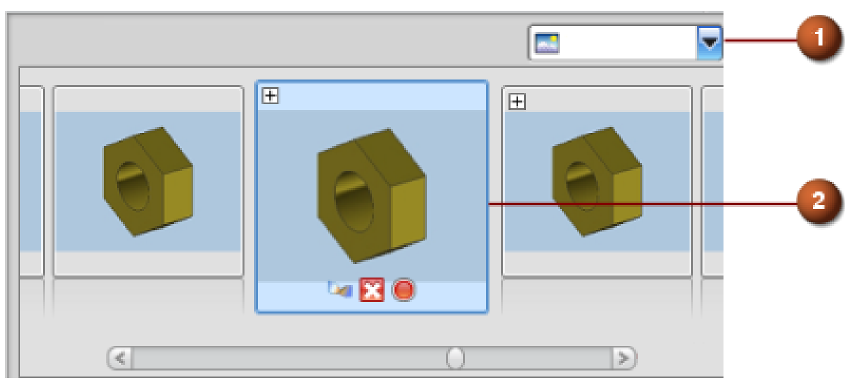
In the **Visual Reporting** tool, view styles are called legend styles.

The following styles are available:

- **Flow List+Tree**: Displays results in a list of thumbnail images that you can scroll through.

  Note

  This style is not available in the **Visual Reporting** tool.

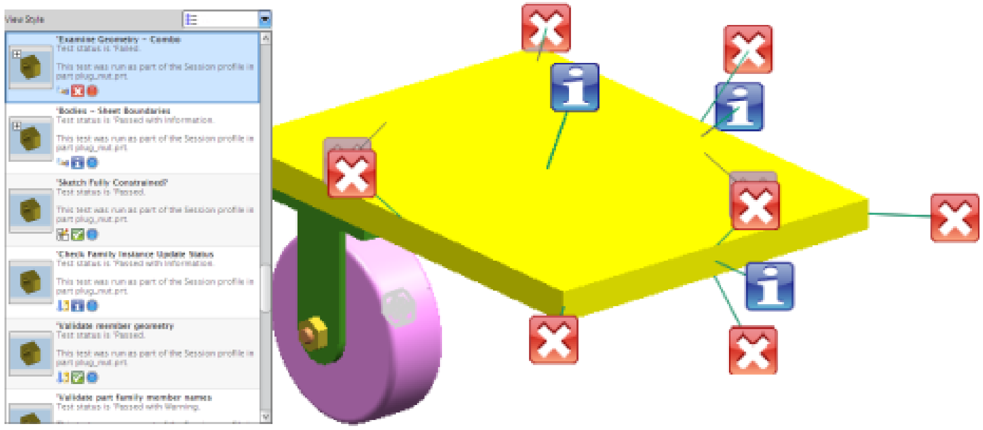


1	**View Style** list
2	Highlighted thumbnail of a tagged object

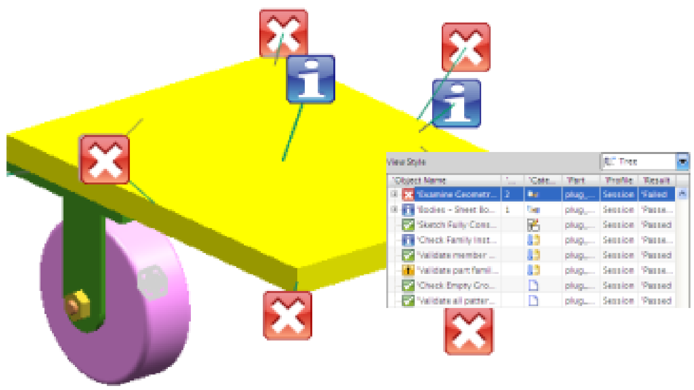
You can:

- Scan the images using the scroll bar. You can also scroll the list by clicking any of the images to the right or left of the centered image.
- Display the tool's results outside of the tool dialog box, in a separate window.
- Stretch the size of the list to make it tall and narrow. The tiles are then arranged vertically.
- Expand and collapse the results for thumbnails that have the + plus sign.
- Focus the view on the selected tag in the graphics window.
- Select objects associated with the tag.

- **Tile List**: Displays results in a tile format that includes a thumbnail and a summary of the result.



- **Tree List**: Displays results in columns. You can click on a column heading to sort results by the values in that column. For example, you can sort results by name, category, part name, profile name, or result status.



- **Info View**: Displays details about one result or piece of information at a time.

  When you first activate an HD3D tool, the view styles are shown in the tool's dialog box. When you double-click a tag or list item, or right-click and select **Show Info View**, an info view window displays the information for that tag or list item.

  You can click the **More Detail** button to see a complete description of the test.

Where do I find it?

Resource bar	Any active HD3D tool.

HD3D tags

What is it?

HD3D tags displayed in the graphics window are a new visual way to interact with information on the 3D model.

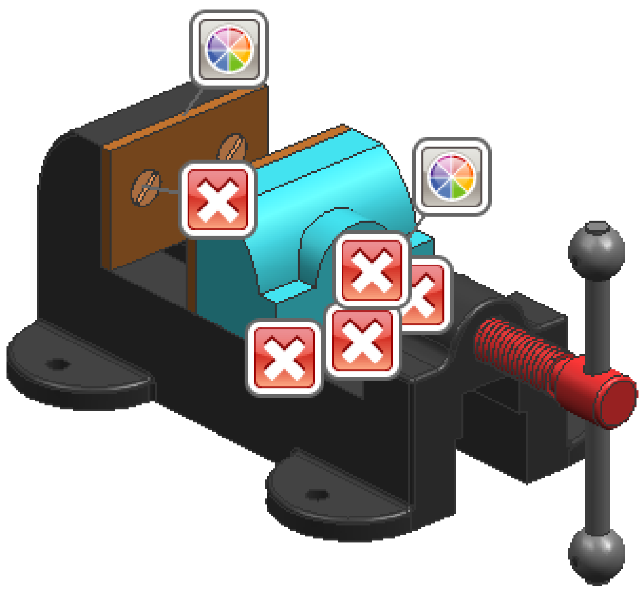
Tags identify objects for which an HD3D tool has information. They present a subset of the results and information that is shown in the HD3D tool dialog box.

Check-Mate tags identify areas with problems or check result information. Visual Reporting tags appear as color wheels.

You can:

- Display tags from every active HD3D tool together in the graphics window.

  In the following graphic, both Visual Reporting tags and Check-Mate tags are displayed on the model.



- Access detailed information and options from the tags.
- Open the **Info View** window for the result or the information the tag represents.
- Remove the tags temporarily from the graphics window. You can restore the tags at any time.

Why should I use it?

Tags help you quickly find and access information about an area of your design.

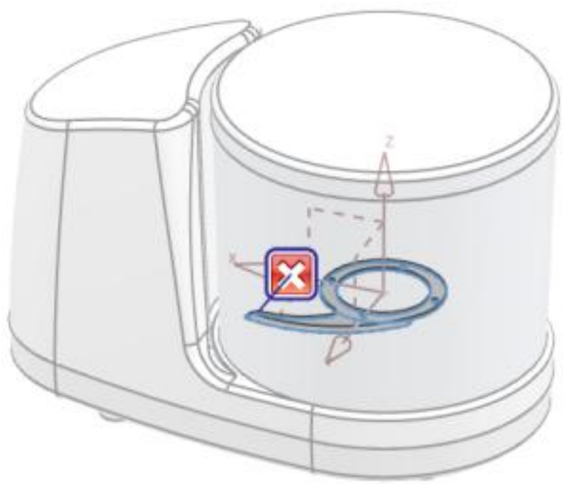
Where do I find it?

Tags are displayed in the graphics window whenever an HD3D tool that uses them is active.

 See-Thru display

What is it?

See-Thru is a new rendering option that displays unimportant facet geometry with translucency. It uses de-emphasis colors to deemphasize unimportant faceted geometry so that important faceted geometry stands out better.



When you apply a visual report that has unmatched components, the See-Thru option is automatically selected and applied to the unmatched components.

You can manually change the See-Thru settings (style or color preferences) while a visual report is active. When you deactivate the report, the See-Thru options return to their default settings.

There are three styles of See-Thru rendering:

 **See-Thru Shell**	Displays less important shaded geometry as a translucent shell with a specified color. Hidden unimportant facet geometry is not displayed. This is the default See-Thru style.
 **See-Thru Original Color Shell**	Displays less important shaded geometry as a translucent shell, preserving the original shaded geometry colors. Hidden unimportant facet geometry is not displayed.
 **See-Thru Layers**	Displays less important shaded geometry as translucent layers with a specified color.

You can change the default edge color and translucency for each style in the **Visualization Preferences** dialog box. You can change the default color only for the **See-Thru Shell** and **See-Thru Layers** styles.

Note

- Only sheet and solid bodies are affected by the **See-Thru** options.
- The translucency percentage is hard-coded for each style.
- When you enable the **See-Thru** rendering option , if shaded edges are displayed, that is the shaded edges have some color other than *none*, then the edges are displayed only for the topmost visible layer of unimportant facet geometry, regardless of the See-Thru style.

Why should I use it?

Use the **See-Thru** option to provide for displays that make it easier to see tags or other information inside the 3D model.

Where do I find it?

See-Thru options

Toolbar	**View→See-Thru** **View→See-Thru Style** list **→See-Thru Shell** /**See-Thru Original Color Shell** /**See-Thru Layers**

Visualization preferences for **See-Thru** options

Menu	**Preferences→Visualization**
Location in dialog box	**Visualization Preferences** dialog box→**Faceting** page→**Session Settings group→See-Thru Settings** subgroup **Visualization Preferences** dialog box→**Color Settings** page→**Part Settings group→De-emphasis Settings** subgroup

HD3D preferences

What is it?

Use the **HD3D Tools** dialog box to set preferences for the HD3D tools.

You can:

- Turn on or off one, or all HD3D tools. When you turn off all HD3D tools, the HD3D tools tab is also removed from the Resource bar.
- Reorder the HD3D tools.
- Select the NX applications in which you want the HD3D tools to appear. You can select the tool or tools you want in the **HD3D Tools** dialog box and then select the applications you want in the

HD3D Tool Properties dialog box. For example, you can have the **Check-Mate** tool show up only in the Modeling application.

Note

The HD3D tools remain active even after you close the **HD3D Tools** tab in the Resource bar.

Where do I find it?

HD3D Tools dialog box

Menu	**Preferences→HD3D Tools**

HD3D Tool Properties dialog box

Menu	**Preferences→HD3D Tools** dialog box→**Properties**
Shortcut menu	In the HD3D Tool Manager, right-click an HD3D tool→**Properties**

HD3D Visual Reporting

What is it?

Use the Visual Reporting tool to display components according to the results of a report that you specify. The report consists of the following:

- A report property, which is used to color or tag the part.
- A report scope, which lets you specify the components used in the report. The visual report is applied to all components in the displayed part, unless the scope specifies differently.

A legend in the **Visual Reporting** dialog box lets you interpret the results and perform actions on groups of components that are displayed with the same color. You can open the legend in a separate window.

When you use a visual report, you can:

- Retrieve and activate an existing visual report.
- Define and activate a new report. Some example reports are included with NX.
- Change the value of any user-defined terms of the scope.
- Specify whether the results are shown as colors, tags, or both on your components.
- Work with NX functions as usual. If you make changes that affect the visual report, you can click **Refresh** to update the results.

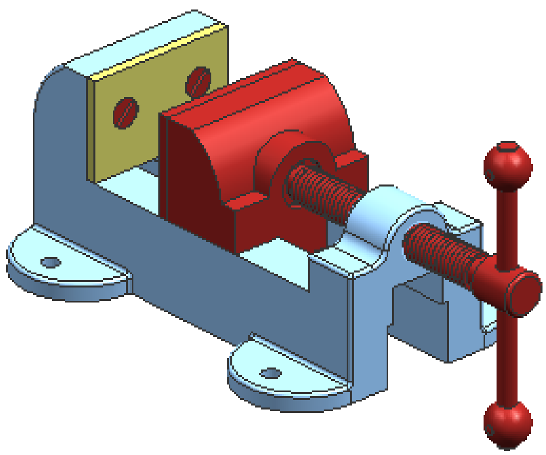
You can use the following properties to build your report:

- Component properties that can be shown as **Assembly Navigator** columns
- Part attributes
- Teamcenter properties that can be retrieved from the Item, Item Revision, or dataset

When you define a visual report, you can:

- Specify the report property and scope.
- Specify the groups that appear in the legend, or you can let the software automatically determine the groups. For example, you can create a visual report based on the **Status** the components have in Teamcenter. You can define a scope term that specifies you are only interested in seeing the status for part files you own.
- Save the visual report as a VPX file for later use.

Results appear in the graphics window, as shown in the following figure.



Visual report with colors only

In the figure above, the visual report displays each component based on the **Position** column in the **Assembly Navigator**. Red components are partially constrained, yellow components are fully constrained, and blue components are fixed.

Why should I use it?

You can use the Visual Reporting tool to quickly visualize information about the design you are working on.

- Coloring components helps you quickly interpret large amounts of information, such as overall trends in the assembly. In the example, you can quickly see that most of the components are partially constrained. If you activate a visual report based on weight, you can easily see if one area is significantly heavier than the others.
- Tagging components provides you with more information than is possible with colors by themselves. For example, when your visual report uses tags to report on weight, you can move your cursor over a tag to see the exact weight of the component.

Where do I find it?

Activate a visual report

Resource Bar	**HD3D Tools**→double-click **Visual Reporting**

Define a visual report

Menu	**File**→**Utilities**→**Define Visual Report**

Define a visual report (alternate options)

Location in dialog box	**Visual Reporting** dialog box→**Report** group→**Define New Report** **Visual Reporting** dialog box→**Report** group→**Edit** the active report, and save under a new name

Open the legend in a separate window

Prerequisite	**Reporting Style** must be set to **Color Objects** or **Color and Tag Objects**.
Shortcut menu	In the **Visual Reporting** dialog box, right-click the legend background→**Send to Window**

Check-Mate

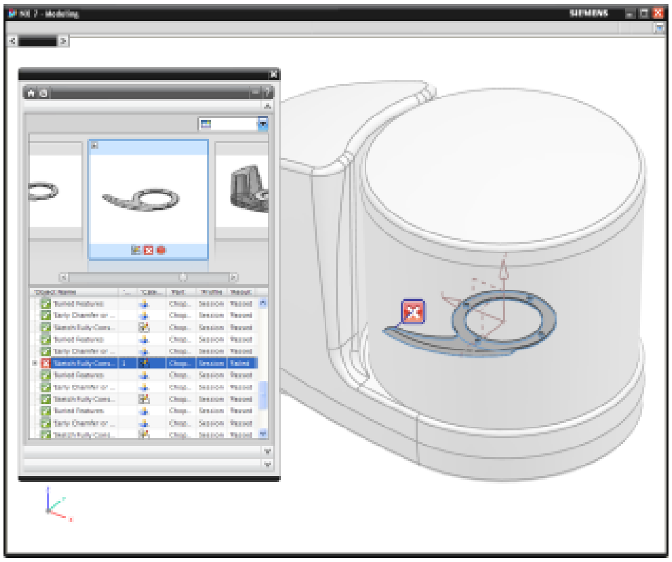
HD3D Check-Mate tool overview

What is it?

Use the HD3D Check-Mate tool to interactively check parts, assemblies, and drawings to ensure that they:

- Follow company design standards.
- Use best practices.
- Meet modeling quality standards.

Check-Mate uses the HD3D visualization tools to graphically tag results directly on the model.



When you use this tool, you can:

- Execute Check-Mate tests interactively as you design your models.
- Display results in a graphical tree list, or using tags on the 3D model.

  See tags clearly on the 3D model using the new See-Thru display commands. Hide tags from the display temporarily by pressing Ctrl+3.
- Access actions from the results list or from the graphic tags. For example, right-click the list item to make the listed part the work part and to select associated features to edit.

Tip

Make the best use of the graphical displays in the HD3D Check-Mate tool by using the full screen display mode. Place the floating information windows on the secondary monitor, if you use dual monitors.

Why should I use it?

You can use the HD3D Check-Mate tool to quickly locate, diagnose, and fix problems.

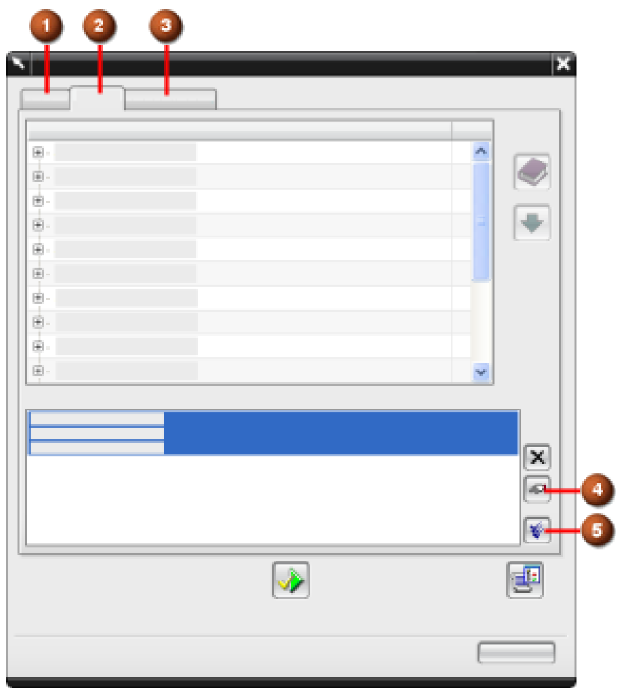
Where do I find it?

Resource bar	**HD3D Tools** tab  →double-click **Check-Mate**.

 Set up Check-Mate tests

What is it?

The **Set Up Tests** command is now available in the HD3D **Check-Mate** Tool dialog box. This button opens the **Check-Mate** dialog box where you can define product validation tests to run.



#	Component	Description
1	**Parts** tab	Lets you select which parts to test, and how many levels of an assembly to include.
2	**Tests** tab	Lets you select tests to perform from lists of provided tests and user-defined profiles.
3	**Run Options** tab	Lets you control log files and other checking options.

#	Component	Description
4	**Customize** button	Lets you define parameters and settings for each test.
5	**Create Temporary Profile** button	Lets you store profiles to save a list of tests with parameters and options already defined.

Why should I use it?

You can set up tests from the same **HD3D Tools** tab that you use to display results.

Where do I find it?

HD3D**Check-Mate** Tool dialog box

Resource bar	**HD3D Tools** tab →double-click **Check-Mate**
Location in dialog box	**Settings** group→**Set Up Tests**

Toolbar and menu locations

Toolbar	**Check-Mate**→**Set up Tests**
Menu	**Analysis**→**Check-Mate**→**Set Up Tests**

Execute HD3D Check-Mate tests

What is it?

The **Execute Check-Mate Tests** command is now available in the HD3D **Check-Mate** Tool dialog box.

You can now:

- Define, run, and review results on the same **HD3D Tools** tab.
- Interact with test results as you create your design.
- Keep the tests active and reuse them on other parts.
- Deactivate Check-Mate with the **Deactivate** button on the HD3D Tools Manager.

Why should I use it?

You can execute tests from the same tab on which you set up tests and review results using the HD3D **Check-Mate** tool.

Where do I find it?

Resource bar	**HD3D Tools** tab →double-click **Check-Mate**
Location in dialog box	**Controls** group→**Execute Check-Mate Tests**

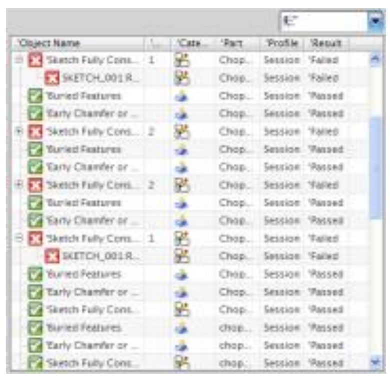
Display Check-Mate results

What is it?

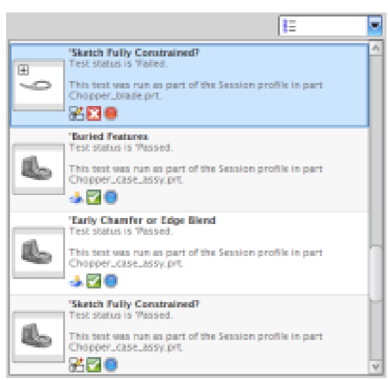
You can now display Check-Mate results graphically to more easily scan for design problems.

You can:

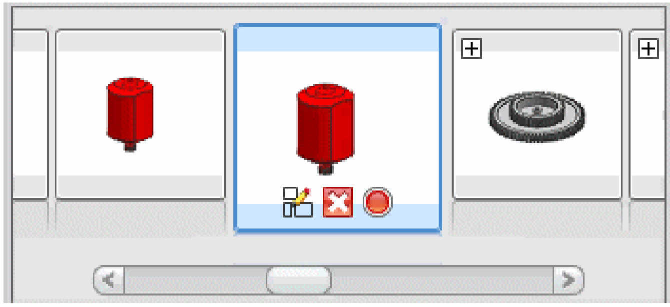
- Display results as a tree list to scan the columns of the results. Click on a column heading to sort results by the values in that column. For example, sort results by name, category, part name, profile name, or result status.



- Display results as tiles to view a thumbnail image and summary information.

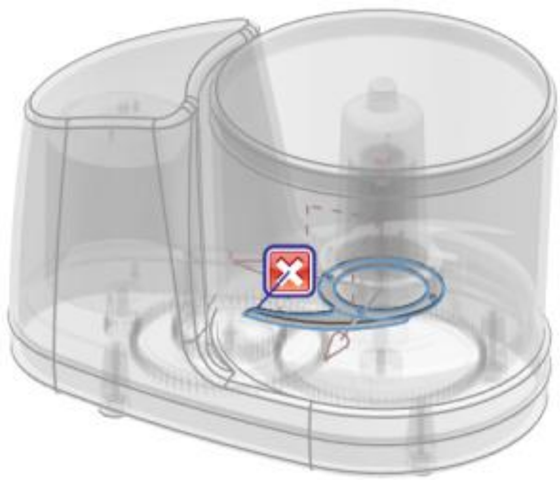


- Display results as a flow list to graphically scan through the thumbnail images at the top of the tree list.

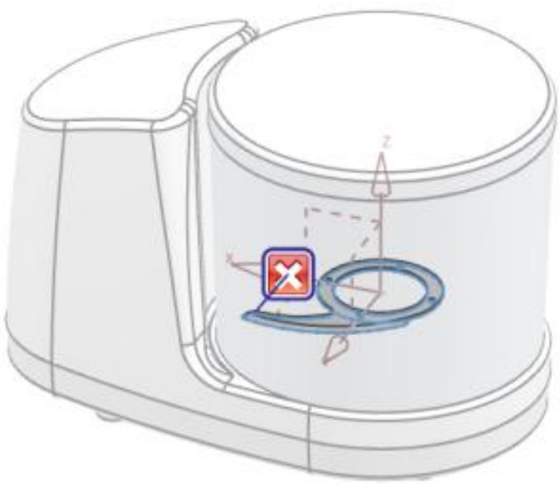


Graphical icons are used to help sort the results in each of the results listings. These icons show a category of result, the pass or fail status, and a priority.

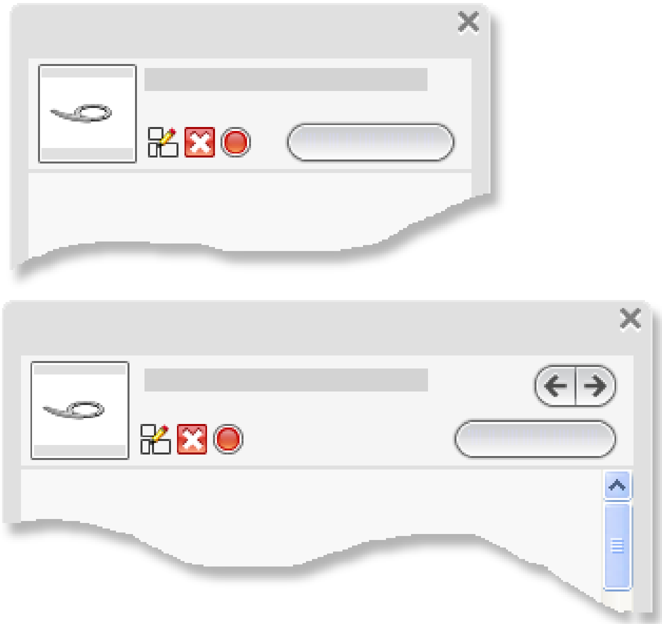
- Locate problems graphically with tags placed on the 3D model in the graphics window.



- Use the See-Thru display types to make tags more visible on a 3D model.



- Double-click a graphical tag or an item in the results list to display the Check-Mate Result window which gives summary information on the test. Click the **More Detail** button to see a complete description of the test.



Why should I use it?

You can graphically scan Check-Mate results using HD3D displays and tags to identify and locate problem areas more quickly than sorting through lists.

Where do I find it?

HD3D **Check-Mate** Tool dialog box

Resource bar	**HD3D Tools** tab  →double-click **Check-Mate**
Location in dialog box	**Results** group

Toolbar and menu locations

Menu	**Analysis→Check-Mate→View Check-Mate Results**
Toolbar	**Check-Mate→View Check-Mate Results** 

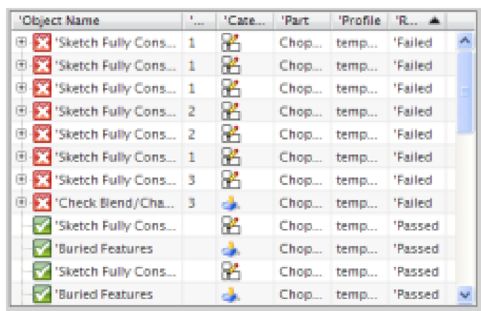
Filter Check-Mate results

What is it?

Use the **Settings** group in the options in the HD3D Check-Mate Tool dialog box to filter Check-Mate results.

For example, the options shown indicate that the software will display failed tests without showing the passed tests, and will filter out tests on parts in your assemblies for which you do not have write permission.

> **Tip**
>
> You can also sort results in the table by clicking on a column heading to sort by that column.



'Object Name	'...	'Cate...	'Part	'Profile	'R...
'Sketch Fully Cons...	1		Chop...	temp...	'Failed
'Sketch Fully Cons...	1		Chop...	temp...	'Failed
'Sketch Fully Cons...	1		Chop...	temp...	'Failed
'Sketch Fully Cons...	2		Chop...	temp...	'Failed
'Sketch Fully Cons...	2		Chop...	temp...	'Failed
'Sketch Fully Cons...	1		Chop...	temp...	'Failed
'Sketch Fully Cons...	3		Chop...	temp...	'Failed
'Check Blend/Cha...	3		Chop...	temp...	'Failed
'Sketch Fully Cons...			Chop...	temp...	'Passed
'Buried Features			Chop...	temp...	'Passed
'Sketch Fully Cons...			Chop...	temp...	'Passed
'Buried Features			Chop...	temp...	'Passed

Why should I use it?

You can filter and sort your test results in the order that you want to fix problems. The results remain available on the HD3D Tools tab as you work with other commands.

Where do I find it?

Resource bar	**HD3D Tools** tab  →double-click **Check-Mate**
Location in dialog box	**Settings** group

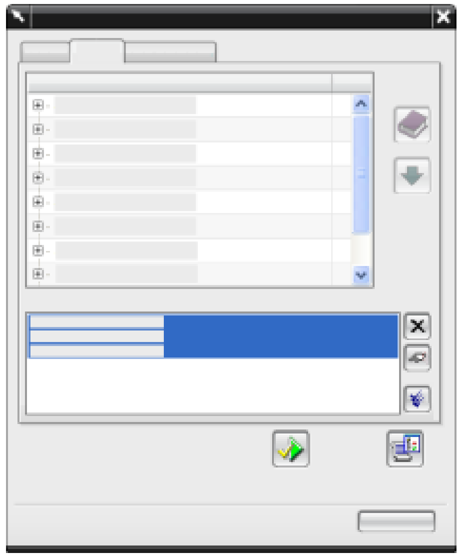
Check-Mate customization

What is it?

There are new ways to customize Check-Mate, depending on the role of the user, and the amount of customization required.

From the HD3D **Check-Mate** Tool dialog box users can:

- Choose from a list of supplied tests as the simplest level of customization.

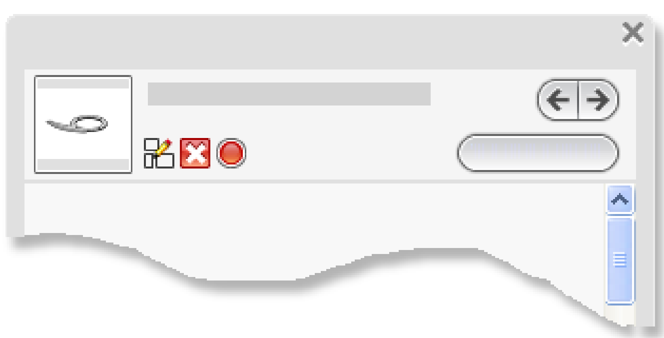


- Edit each test to modify test parameters.
- Store a list of chosen tests as a temporary profile. Users can edit a profile to modify the list of tests, or to modify parameters from each test.

Test authors can customize Check-Mate tests to do the following:

- Create and store global profiles from the **Check-Mate Author Tools** dialog box.
- Use Knowledge Fusion Interactive Class Editor to author tests to create a company profile to enforce company standards.
- Customize the bit-mapped image used to show the result category.

- Modify the test information in the Check-Mate Result window.



- Modify the description, which is used for the node’s tooltip and for the text in the Tile results view list.

Why should I use it?

You can customize Check-Mate test profiles to efficiently reuse the same sets of tests on multiple parts and to enforce company standards.

Where do I find it?

HD3D **Check-Mate** Tool dialog box

Resource bar	**HD3D Tools** tab  →double-click **Check-Mate**
Location in dialog box	**Settings** group, **Set Up Tests** 

Check-Mate Author Tools dialog box

Menu	**Analysis**→**Check-Mate**→**Author Tests**
Toolbar	**Check-Mate**→**Author Tests** 

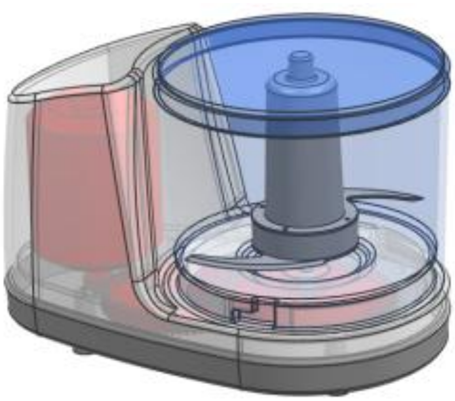
For documentation on customizing, find the instructions distributed with the Check-Mate software.

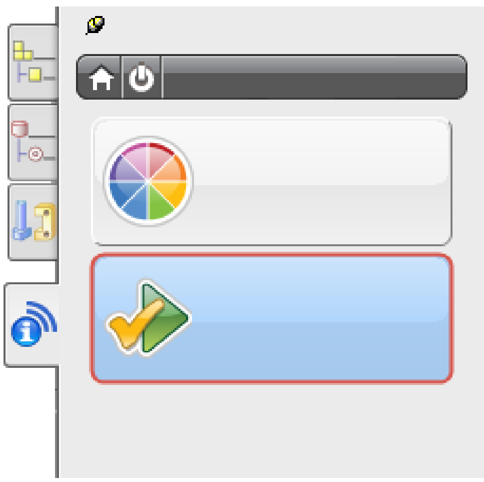
HD3D Check-Mate workflow

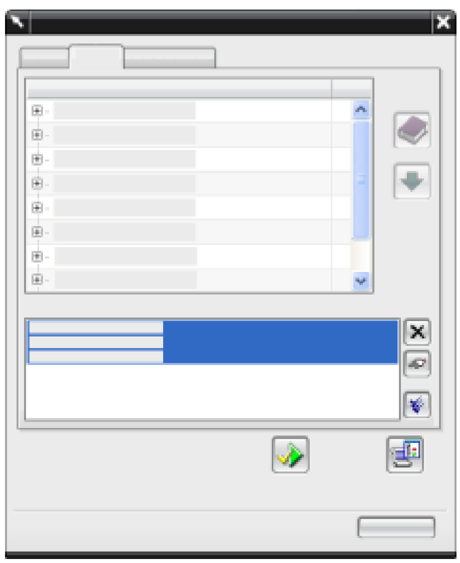
What is it?

Depending on your work role, there are different ways to use Check-Mate.

A typical designer workflow is shown below.

	Start with an open part, assembly, or drawing being designed.
	On the **HD3D Tools** tab, double-click **Check-Mate**.





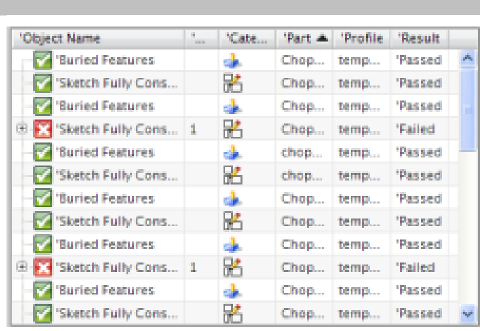
In the **Settings** group, click **Set Up Tests**  .

On the **Parts** tab, select the part file and the levels of the assembly to check.

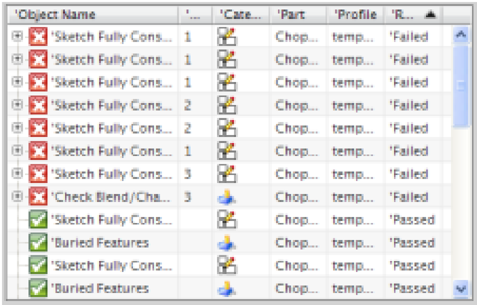
Select the tests to run in the **Categories** list. Select from standard tests or from your local company profiles.

Click **Add To Selected** to move the selected tests to the **Chosen Tests** list.

Click **Close**.

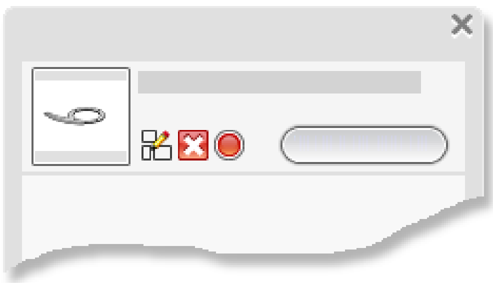


To run the chosen tests, in the **Controls** group, click **Execute Check-Mate Tests**  .

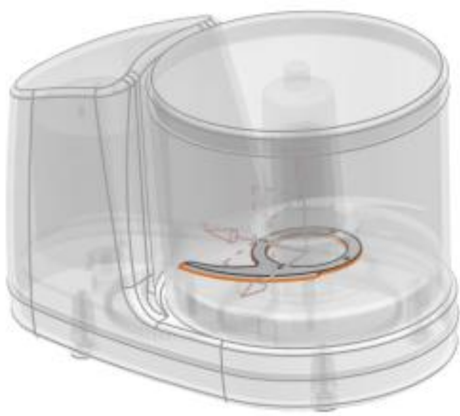


To sort by the result status, in the **Results** group, click on the **Result** column heading.

To filter results, use the options in the **Settings** group.

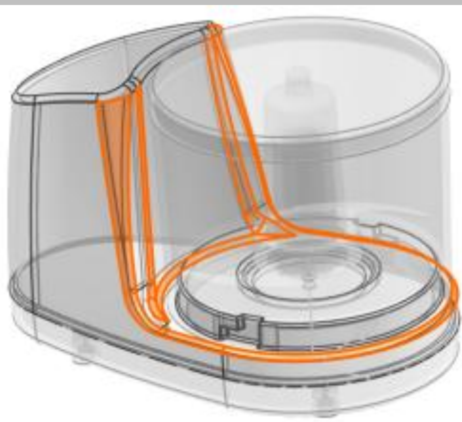


To review any problem areas, click on the result objects in the **Results** group. Double-click to see information in the Check-Mate Result window.



To edit a feature in an assembly component follow these steps.

1. Right-click on an error and select **Make Work Part**.
2. Right-click and select **Select Associated Objects**.
3. Edit the feature using the **Part Navigator**.



Continue to work with the model and review tests.

Press Ctrl+3 to temporarily hide graphic tags.

Open other parts to run the same tests.

Deactivate Check-Mate when finished using **Deactivate** at the top of the **HD3D Tools Manager**.

Chapter 4: Gateway

Curve Continuity

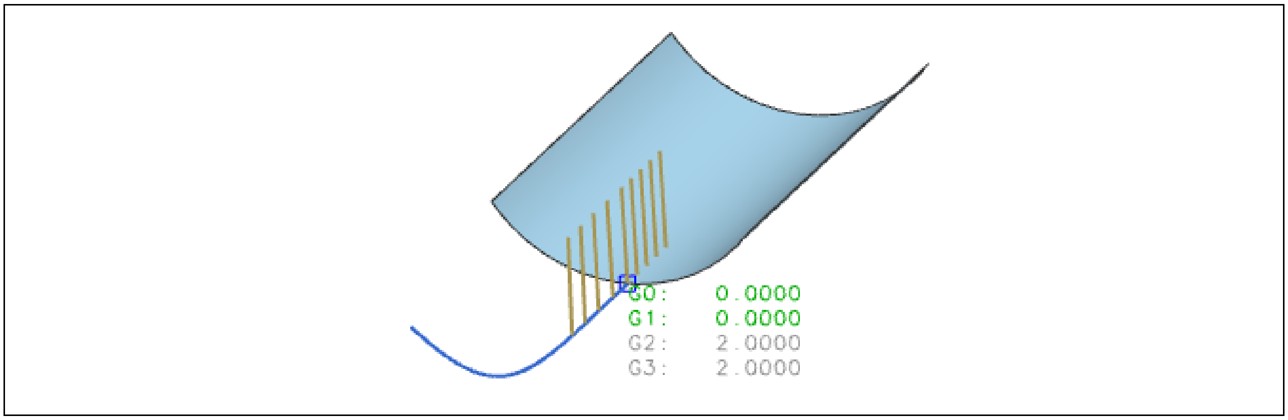
What is it?

Use the **Curve Continuity** command to evaluate the continuity between a curve and a reference object. A reference object can be a face, an edge, a curve, or a datum plane.

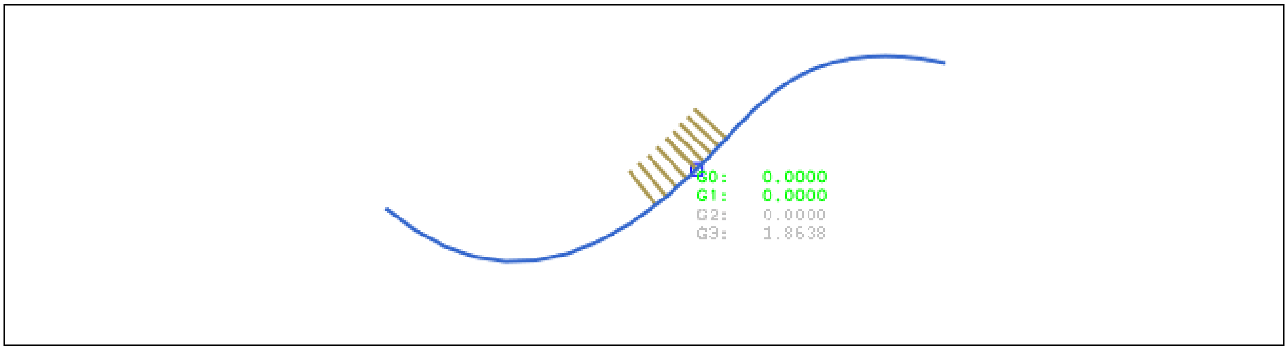
You can calculate G0, G1, G2, and G3 continuity values between the end point of the curve and the nearest point on the reference object.

An associative curve continuity analysis object is created with a label at the end point of the curve.

How the continuity constraints are displayed depends on the continuity types specified in the **Curve Continuity** dialog box. If the deviation for a continuity type exceeds the modeling tolerances, the continuity type is shown in red.

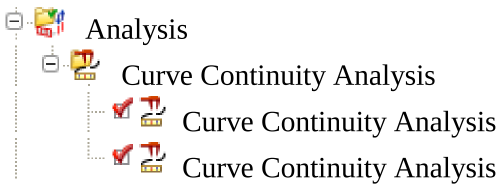


Curve continuity analysis between a curve and a face



Curve continuity analysis between two curves

The associative curve continuity analysis objects are listed in the main panel of the **Part Navigator**. You can select or clear the check boxes to show or hide the labels in the graphics window.



Why should I use it?

You can check curve deviations, such as variation in position, tangency, curvature, and acceleration between faces, curves, edges, or the normal of a datum plane.

This command is useful in the surfacing workflow, where you must verify the continuity between curves that are used as inputs to create surfaces.

Where do I find it?

Application	Modeling
Toolbar	**Analyze Shape→Curve Continuity Analysis**
Menu	**Analysis→Curve→Continuity**

Curve Analysis

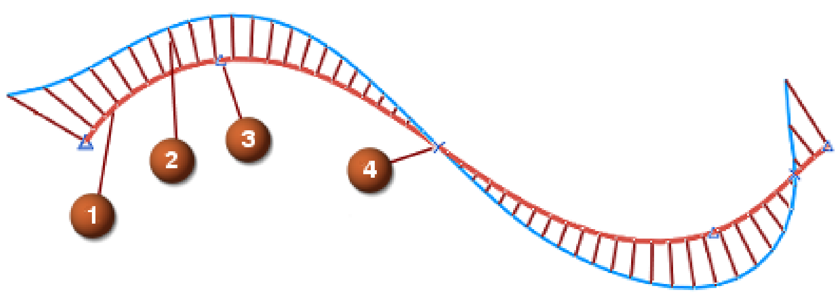
What is it?

Use the **Curve Analysis** commands to analyze the shape of an edge or a curve.

For a selected curve or edge, you can:

- Display the curvature comb. You can specify the curvature comb parameters such as the number of needles, needle scale, scale factor, and so on.
- Create and display curvature peak points and curvature inflection points.

Curve analysis objects dynamically update as you change the curves on which they are based, or when you change the analysis parameters.



1. Curve
2. Curvature comb
3. Peak point
4. Inflection point

The associative curve analysis objects are listed in the main panel of the **Part Navigator**. You can select or clear the check boxes to show or hide the curvature comb, peak point, and inflection point symbols in the graphics window as required.

- Analysis
- Curve Analysis
  - Curve Analysis

You can specify the color and line fonts for the needle and cap line of the curve analysis objects in the **Edit Object Display** and **Object Preferences** dialog box. For more information, see Curve Analysis Display options.

Why should I use it?

You can analyze and display the curve curvature, and verify that the curve is smooth and does not include unintended inflections.

Where do I find it?

Application	Modeling
Toolbar	**Analyze Shape→Curve Analysis** →**Curve Analysis – Combs** / **Curve Analysis – Peaks** / **Curve Analysis – Inflections**
Menu	**Analysis→Curve→Curve Analysis**

Curve Analysis Display options

What is it?

The following options are available in the **Edit Object Display** and **Object Preferences** dialog box:

Needle

A color swatch lets you specify the color of the needles.

The line font list lets you specify the line font of the needles.

Cap

Select this check box to display the cap line.

A color swatch lets you specify the color of the cap line.

The line font list lets you specify the line font of the cap line.

Why should I use it?

Use these options to specify the color and line fonts for the needles and cap lines of curve analysis objects.

Where do I find it?

Toolbar	**Utility**→**Edit Object Display** 
Menu	**Edit**→**Object Display** **Preferences**→**Object**
Location in dialog box	**Edit Object Display** dialog box→**Analysis** tab→**Curve Analysis Display** group **Object Preferences** dialog box→**Analysis** tab→**Curve Analysis Display** group

Combs Options, Peaks Options, and Inflections Options

What is it?

The following commands are no longer available in the **Analysis** menu:

- **Combs Options**
- **Peaks Options**
- **Inflections Options**

The parameters for controlling curvature combs, curvature peak points, and curvature inflection points are available in the **Curve Analysis** dialog box. For more information, see Curve Analysis.

Chapter 5: Design

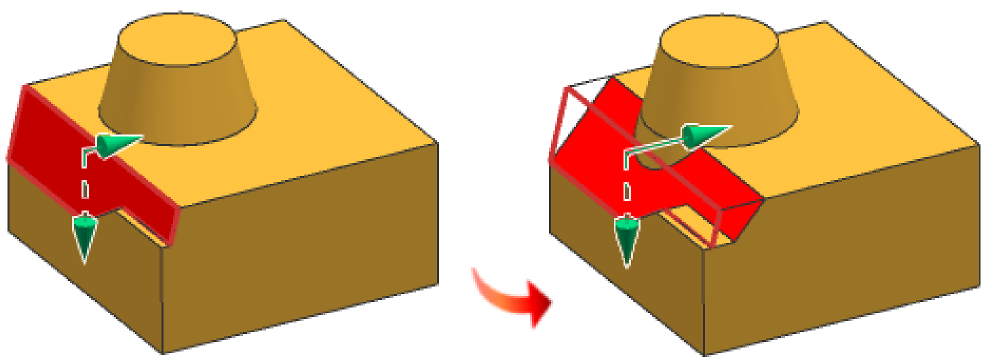
Modeling

Synchronous Modeling with synchronous technology

 **Resize Chamfer**

What is it?

Use the **Resize Chamfer** command to change the size of a chamfered edge regardless of its feature history.



Why should I use it?

You can edit the size of an angled face so that it has the one of the following parameters:

- Symmetric Offset
- Asymmetric Offset
- Offset and Angle

Where do I find it?

Application	Modeling
Prerequisite	A face must be labeled as a chamfer using the **Label Chamfer** command if it cannot be recognized as a chamfer.

Toolbar	**Synchronous Modeling→Resize Chamfer**
Menu	**Insert→Synchronous Modeling→Chamfer→Resize Chamfer**

Label Chamfer

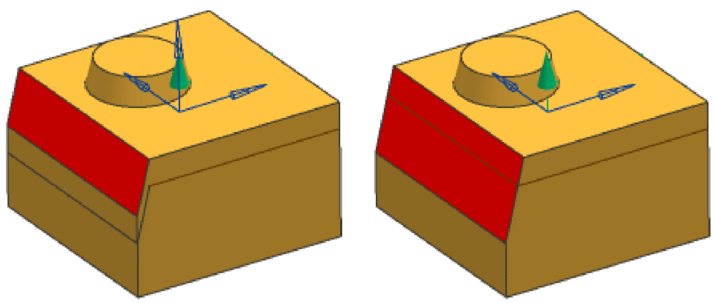
What is it?

Use the **Label Chamfer** command to label an angled face as a chamfer. The angled face can be on the NX model or on an imported solid.

Use this command when you want the following commands to recognize an angled face as a chamfer:

- **Resize Chamfer**
- **Move Face**

In the model on the left the highlighted face is labeled as a chamfer.



Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling→Label Chamfer**
Menu	**Insert→Synchronous Modeling→Chamfer→Label Chamfer**

Optimize Face

What is it?

Use the **Optimize Face** command to simplify surface types, merging faces, improving edge accuracy, and recognizing blends.

Why should I use it?

Use this command on models imported into NX to convert B-surface faces to analytic faces.

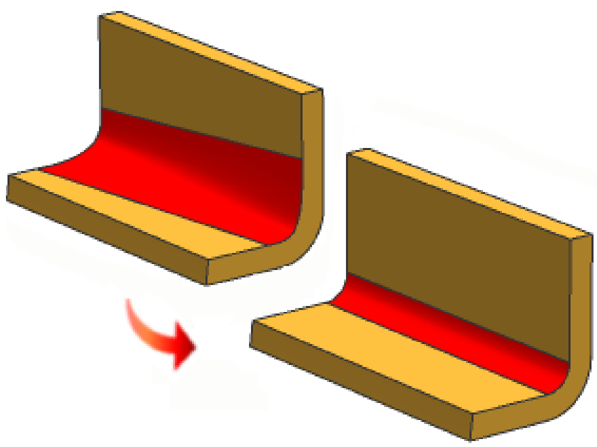
Where do I find it?

Application	Modeling
Prerequisite	You must be in History-free mode
Toolbar	**Synchronous Modeling→Optimize Face** 
Menu	**Insert→Synchronous Modeling→Optimize→Optimize Face**

 **Replace Blend**

What is it?

Use the **Replace Blend** command to convert B-surface faces that appear like blends to a replace blend feature.



Why should I use it?

Use this command on models imported into NX to convert B-surface faces to rolling ball type blend faces.

You can inherit the blend radius from the selected face, or you can enter a radius.

Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling→Replace Blend**
Menu	**Insert→Synchronous Modeling→Optimize→Replace Blend**

Create Feature – enhancement

What is it?

You can now turn on or off creating features when you select or clear the **Create Feature** option while working in History-free mode.

You can use this option with the **Hole**, **Edge Blend**, and **Chamfer** commands.

Why should I use it?

Use this option to edit the feature parameters later.

With the **Hole** command, the parameters of the hole are maintained in relation to the planar face when you select the **Create Feature** option. For example, if the planar face is moved, the bottom of the hole also moves.

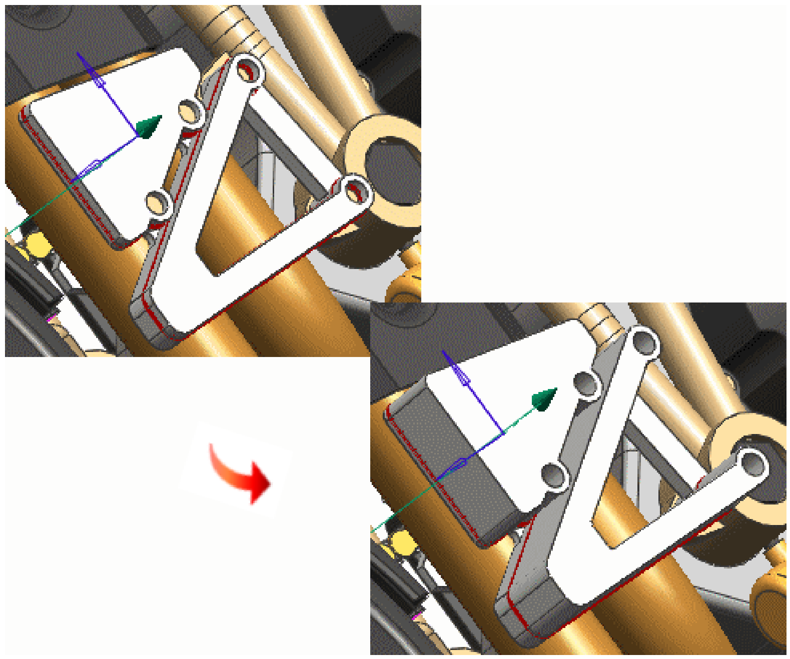
Where do I find it?

Application	Modeling
Prerequisite	You must be in History-free mode
Toolbar	**Feature→Hole** **Feature Operation→Edge Blend** **Feature Operation→Chamfer**
Menu	**Insert→Design Feature→Hole** **Insert→Detail Feature→Edge Blend** **Insert→Detail Feature→Chamfer**
Location in dialog box	**Settings** group→**Create Feature**

Move Face – enhancement

What is it?

When you use the **Move Face** command, you can select faces in components other than the work part within an assembly.



Why should I use it?

You can move multiple component faces of an assembly at the same time.

Where do I find it?

Entire Assembly selection option

Application	Modeling
Prerequisite	All parts must be in History-free mode
Selection bar	**Selection Scope→Entire Assembly**

Move Face command

Application	Modeling

Toolbar	**Synchronous Modeling→Move Face**
Menu	**Insert→Synchronous Modeling→Move Face**

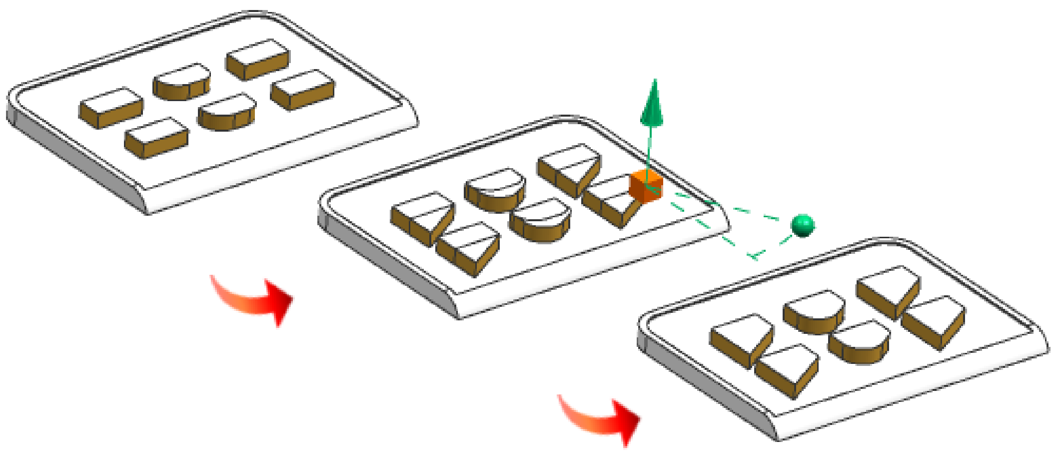
Pattern Face – enhancement

What is it?

When you use the **Pattern Face** command while in History-free mode, a feature that you can edit is created in the **Part Navigator**.

When you use the **Move Face**, **Pull Face**, or **Offset Region** commands on any instance of the pattern, all instances of the pattern are updated.

Other features, for example, **Edge Blend**, **Chamfer**, and **Hole**, that are listed in the Part Navigator, and applied to individual edges or faces in an instance of the pattern, are also updated when you edit the pattern.



Why should I use it?

You can maintain the pattern behavior in history-free modeling.

Where do I find it?

Application	Modeling
Prerequisite	You must be in History-free mode The pattern must be created in NX 7.0 or later
Toolbar	**Synchronous Modeling→Pattern Face** .
Menu	**Insert→Synchronous Modeling→Pattern Face**

Make Fixed

What is it?

Use the **Make Fixed** command to add a fixed constraint to selected faces.

The constraint is also listed in the **Part Navigator**.

The fixed constraint appears in the **Saved** list in all **Relate** command dialog boxes. When you create a relation between a face with a fixed constraint and another face, you can turn off the fixed constraint.

Why should I use it?

Use **Make Fixed** on a face if you want to prevent the face from changing its position when surrounding faces are changed by other synchronous modeling commands.

Where do I find it?

Application	Modeling
Prerequisite	You must be in History-free mode
Toolbar	**Synchronous Modeling→Make Fixed**
Menu	**Insert→Synchronous Modeling→Relate→Make Fixed**

Saved list

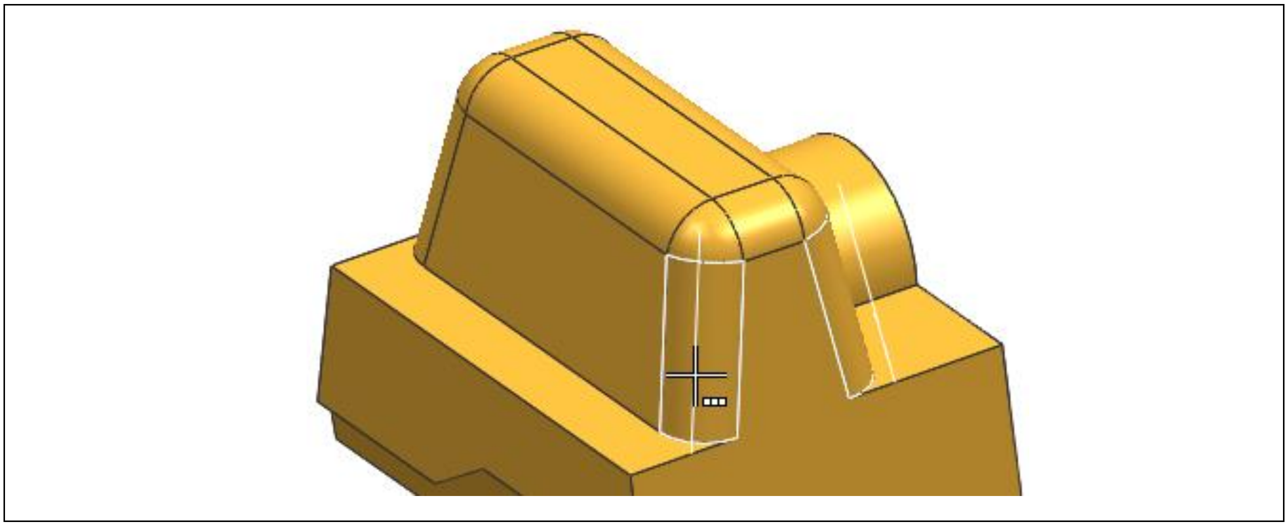
Application	Modeling
Prerequisite	You must be in History-free mode, in one of the **Relate** command dialog boxes
Menu	**Insert→Synchronous Modeling→Relate**
Location in dialog box	[Relate command dialog box]→**Motion** group→**Results** tab→**Saved** list

Dimension to Virtual Edge – enhancement

What is it?

You can now use either the **Linear Dimension** or the **Angular Dimension** command to create a dimension to the virtual edge at the intersection of two faces separated by a blend.

The virtual edge is highlighted as you move the cursor over an edge blend, to indicate that the virtual edge can be selected.



Angular dimension

Where do I find it?

Dimension commands

Application	Modeling
Toolbar	**Synchronous Modeling→Linear Dimension**  **Synchronous Modeling→Angular Dimension** 
Menu	**Insert→Synchronous Modeling→Dimension→Linear Dimension** **Insert→Synchronous Modeling→Dimension→Angular Dimension**

Dimension – enhancement

What is it?

You can use the **Lock Dimension** option to define a static relationship between edges and faces when you create linear, angular, or radial dimensions.

The locked dimension appears in the **Saved** list in all **Relate** command dialog boxes. When you create a relation between a face with a locked dimension and another face, you can turn off the locked dimension.

Why should I use it?

Use the **Lock Dimension** option to prevent the face from changing its position when surrounding faces are changed by other synchronous modeling commands.

Where do I find it?

Lock Dimension option

Application	Modeling
Prerequisite	You must be in History-free mode
Toolbar	**Synchronous Modeling→Linear Dimension** **Synchronous Modeling→Angular Dimension** **Synchronous Modeling→Radial Dimension**
Menu	**Insert→Synchronous Modeling→Dimension→Linear Dimension** **Insert→Synchronous Modeling→Dimension→Angular Dimension** **Insert→Synchronous Modeling→Dimension→Radial Dimension**
Location in dialog box	**Settings** group→**Lock Dimension**

Saved list

Application	Modeling
Prerequisite	You must be in History-free mode, in one of the **Relate** command dialog boxes
Menu	**Insert→Synchronous Modeling→Relate**
Location in dialog box	[Relate command dialog box]→**Motion** group→**Results** tab→**Saved** list

Show Related Face

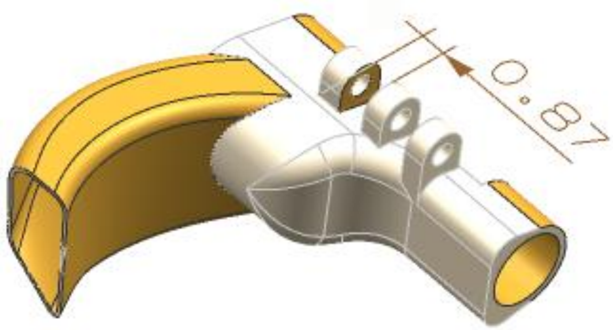
What is it?

Use the **Show Related Face** command to highlight and review relations that exist on faces.

Use this command to show faces with the following relations:

- Fixed
- Linear Dimension
- Angular Dimension
- Radial Dimension
- Offset Relations

For the three dimension commands this relation is saved when the **Lock Dimensions** option is selected.



Why should I use it?

You can:

- Highlight relations that will not change when surrounding faces are changed by other synchronous modeling commands.
- Delete the highlighted relations.

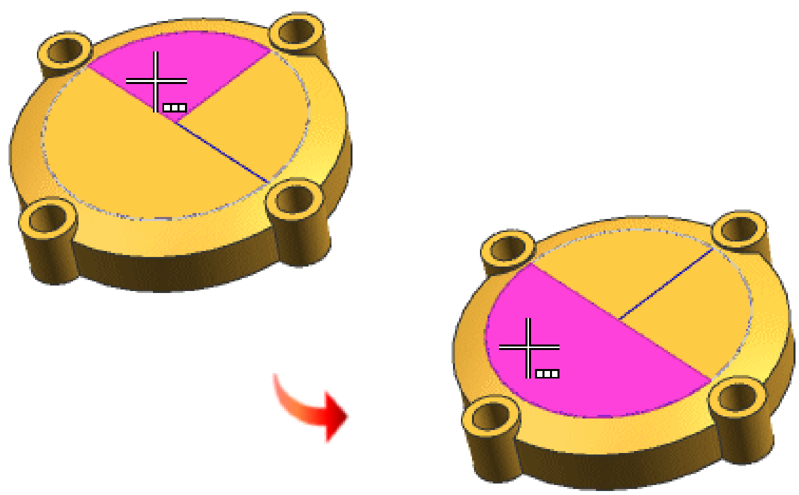
Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling→Show Related Face** 
Menu	**Insert→Synchronous Modeling→Relate→Show Related Face**

Region Boundary Faces – enhancement

What is it?

When using the **Pull Face** and **Offset Region** commands, you can select the portion of a face using the **Region Boundary Faces** selection option. These regions are determined by existing edges and curves that lie on the face.



Why should I use it?

This selection option helps you select portions of a face without using the **Divide Face** command.

Where do I find it?

Region Boundary Faces selection option

Application	Modeling
Prerequisite	You must be in History-free mode
Toolbar	**Selection Bar→Face Rule→Region Boundary Faces**

Pull Face and **Offset Region** commands

Application	Modeling
Toolbar	**Synchronous Modeling→Pull Face**  **Synchronous Modeling→Offset Region** 

Menu	**Insert→Synchronous Modeling→Pull Face** **Insert→Synchronous Modeling→Offset Region**

Face Finder – enhancement

What is it?

You can use the **Select Offset** option in the **Face Finder** group to select faces that appear to be offset from each other.

This option is also available in the dialog boxes for the following commands:

- **Move Face**
- **Offset Region**
- **Resize Face**
- **Cut Face**
- **Copy Face**
- **Mirror Face**
- **Relate** commands
- **Dimension** commands
- **Change Shell Thickness**
- **Group Face**

Why should I use it?

You can use the **Select Offset** option to simplify the selection of faces that appear to be offset from each other.

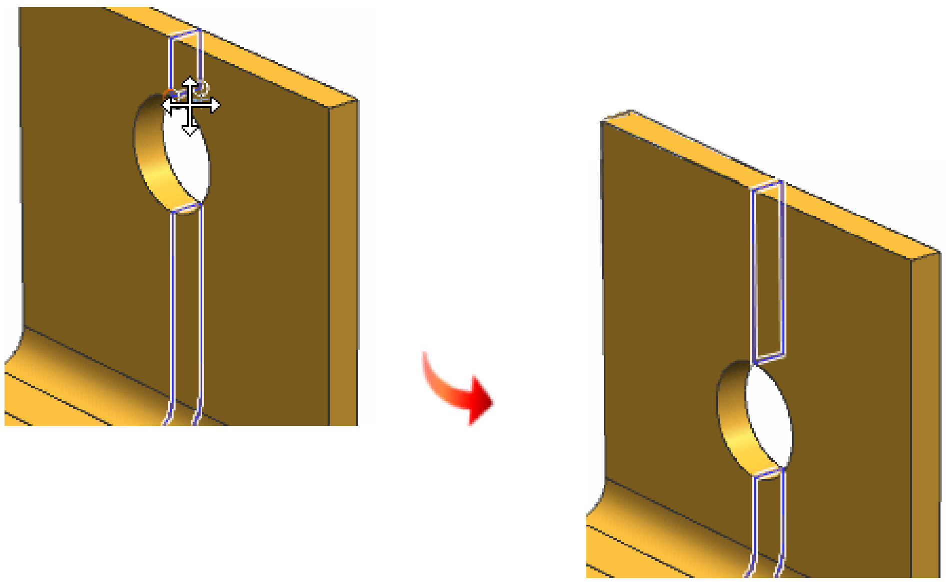
Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling**→[command button]
Menu	**Insert→Synchronous Modeling**→[command]
Location in dialog box	[command dialog box]→**Face Finder** group→**Settings** tab→**Select Offset**

Cross Section Edit – enhancement

What is it?

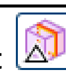
You can move the center of a hole by dragging just one of its cross section curves created by the **Cross Section Edit** command.



Why should I use it?

This allows you to change the center position of a hole while maintaining its original diameter.

Where do I find it?

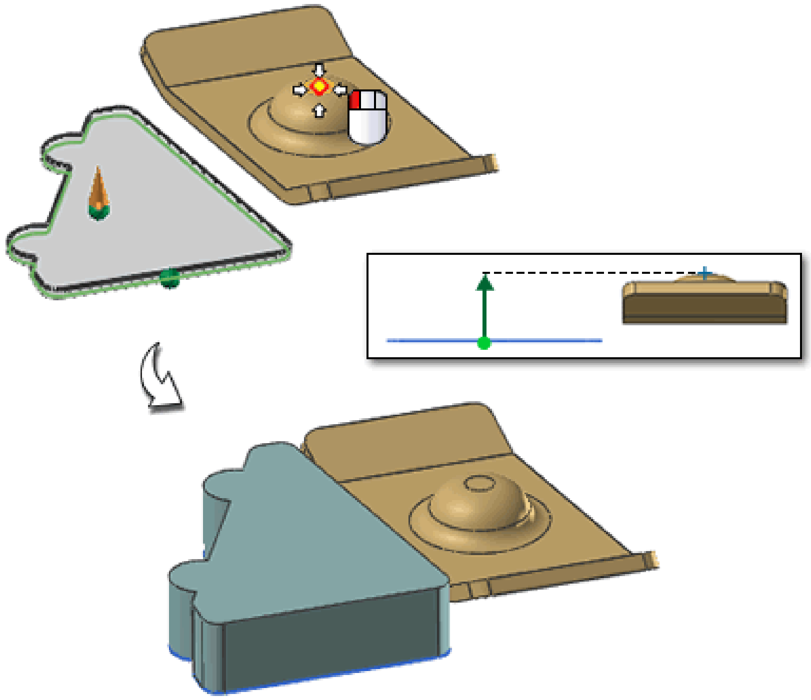
Application	Modeling
Prerequisite	You must be in History-free mode
Toolbar	**Synchronous Modeling→Cross Section Edit** 
Menu	**Insert→Synchronous Modeling→Cross Section Edit**

Snap to an object without associativity

What is it?

For the **Extrude** and **Revolve** commands, the new **Snap to Object** option lets you define a **Start** or **End** distance value by selecting a point, a planar face, or a datum plane. The software calculates the distance along the extrude vector or the revolve arc, and sets the value. There is no associativity to the point, face, or datum plane.

The **Snap to Object** option works in the context of an assembly. For example, in the following assembly, you can use the **Snap to Object** option to extrude the profile to the center point of the dimple.



Note

- For an extrude operation, if you use a planar face or datum plane as the “Snap to” object, Siemens PLM Software recommends that the face or datum be normal to the extrude vector.
- For a revolve operation, if you use a planar face or datum plane as the “Snap to” object, Siemens PLM Software recommends that the face or datum be normal to the revolve arc where the face or plane intersects the arc.

Why should I use it?

The **Snap to Object** option makes it easy to calculate the **Start** or **End** distance to a point, planar face, or datum plane. Use this option when associativity to the limiting point, face, or plane is not important.

Where do I find it?

Application	Modeling
Prerequisite	For the **Extrude** or **Revolve** commands, the **Start** or **End** limit must be set to **Value**.
Graphics window	Right-click a **Start** or **End** handle and choose **Snap to Object**.

Automatically make datums and sketches internal

What is it?

You can set two new Modeling preferences or customer defaults that automatically make

- Datums internal to their child sketches.
- Sketches internal to their child features.

The software automatically internalizes datums or sketches only when you create a child sketch or a child feature. You can still make datums or sketches external using the existing **Part Navigator** commands. The new settings will not cause datums and sketches to be internalized again when you edit the feature.

Why should I use it?

Internal datums and sketches do not appear in the **Part Navigator**, so you can use these settings to shorten the Model history for a part.

Where do I find it?

Modeling preference

Application	Modeling
Menu	**Preferences**→**Modeling**
Location in dialog box	**Modeling Preferences**→**General** tab→**Automatically Make Datums Internal to Child Sketch**/**Automatically Make Sketch Internal to Child Feature**

Customer Default

Menu	**File**→**Utilities**→**Customer Defaults**
Location in dialog box	**Modeling**→**General**→**Miscellaneous**→**Automatically Make Datums Internal to Child Sketch**/**Automatically Make Sketch Internal to Child Feature**

Modeling Preferences

What is it?

The following options are removed from the **Modeling Preferences** dialog box→**Analysis** tab→**Curve Curvature Display** group:

- **Display**: **Curvature**, **Radius of Curvature**

  Style: **Linear**, **Logarithmic**

  Inside , **Outside**

These options are now available in the **Curve Analysis** dialog box. For more information, see Curve Analysis.

- **Show Cap Line**

  This option is now available in the **Edit Object Display** and **Object Preferences** dialog box.

  For more information, see Curve Analysis Display options.

Where do I find it?

Curve Analysis options

Menu	**Analysis→Curve→Curve Analysis**
Toolbar	**Analyze Shape→Curve Analysis** 
Location in dialog box	**Curve Analysis** dialog box→**Settings** group→**Needle Direction**, **Calculation Method**, **Scaling Method**

Curve Analysis Display options

Toolbar	**Utility→Edit Object Display** 
Menu	**Edit→Object Display** **Preferences→Object**
Location in dialog box	**Edit Object Display** dialog box→**Analysis** tab→**Curve Analysis Display** group→**Cap** **Object Preferences** dialog box→**Analysis** tab→**Curve Analysis Display** group→**Cap**

Shape Studio

Specify editor to edit a spline

What is it?

You can specify to use either **X-Form** or **Studio Spline** as the default editor when you double-click a spline.

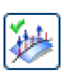
Why should I use it?

If you want to control degrees or segments, use **X-Form** as the editor when you double-click a spline.

If you want to edit spline constraints, use **Studio Spline**.

Where do I find it?

Application	Modeling, Shape Studio
Prerequisite	The setting is effective only in **Shape Studio**
Menu	**Preferences→Modeling** **Files→Utilities→Customer Defaults**
Location in dialog box	**Modeling Preferences→Freeform** tab→**Default Action on Spline** **Customer Defaults→Modeling→FreeForm Modeling→General** tab→**Default Action on Spline**

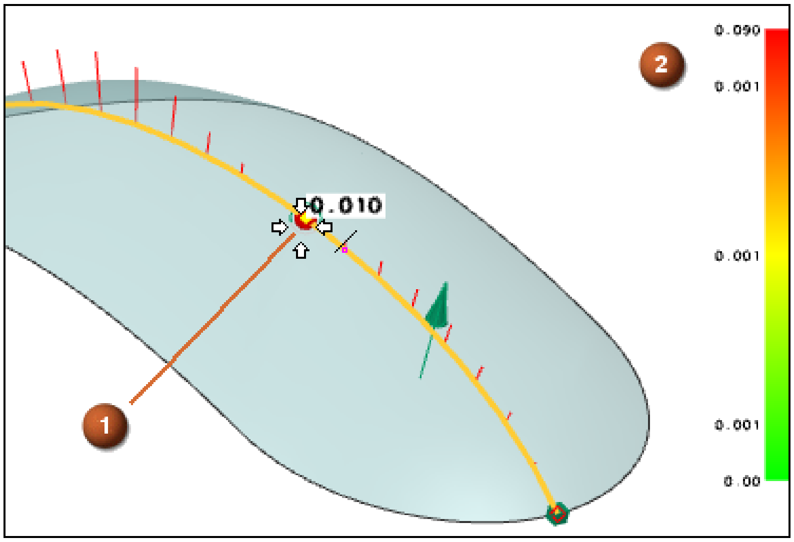


Deviation Gauge

What is it?

The **Deviation Gauge** command has the following enhancements:

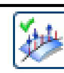
- The **Absolute Value** option lets you control values visible in the label.
- The **Dynamic Label** option lets you see the deviation at the location of a handle, as you drag it across the selected source geometry.
- A color legend is available in the graphics window to show the difference between minimum and maximum deviations.
- Color mapping between a facet body and a face appear on the face.
- Results of the deviation analysis are visible in the dialog box, and include the total number of samples, the number of samples inside both inner and outer tolerance, and the number of samples out of tolerance.

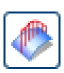


1 **Dynamic Label**

2 Color legend

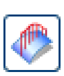
Where do I find it?

Application	Shape Studio
Toolbar	**Analyze Shape→Deviation Gauge** 
Menu	**Analysis→Deviation→Gauge**
Location in dialog box	**Deviation Gauge→Plot** group→**Absolute Value** **Deviation Gauge→Label** group→**Dynamic Label**

 Section Analysis and Grid Analysis

What is it?

The functionality in the **Section Analysis** command is divided between two separate commands: **Section Analysis** and **Grid Analysis**.

 **Section Analysis** command

- You can analyze shapes by cutting sections on selected objects.
- You can control the orientation of the cutting plane.
- You can create section curves separately, without also having to create section analysis objects.
- You can specify section placement using these options:

  - **Uniform** — Cuts a section based on the number of sections or the spacing between the sections.
  - **Through Points** — Cuts sections through user specified points.
  - **Between Points** — Cuts sections between user specified points, adjusted for the number of sections or the spacing of sections.
  - **Interactive** — Cuts sections on the selected objects by drawing each section through two defining points in the graphics window. Sections are cut in the view direction.
- The orientation of curvature needles for an Isoparametric section now always measures and displays the surface curvature of the selected faces, and not the section curve curvature as in prior releases.

Grid Analysis command

- You can analyze shapes by defining a grid to cut sections on selected objects.

Preferences for the curvature display **Needle Direction**, **Calculation Method**, and **Scaling Method** options have been moved from **Modeling Preferences** to both the **Section Analysis** and **Grid Analysis** dialog boxes.

Why should I use it?

Splitting section and grid analysis functionality into separate commands improves their interaction and efficiency for Class A workflows.

Where do I find it?

Application	Shape Studio
Toolbar	**Analyze Shape→Section Analysis** **Analyze Shape→Grid Analysis**
Menu	**Analysis→Shape→Section** **Analysis→Shape→Grid Analysis**

X-Form

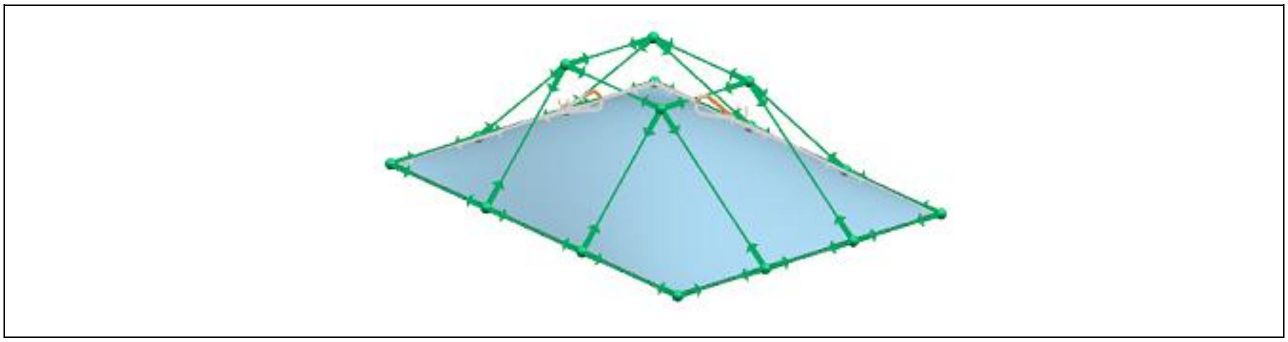
What is it?

The new simplified **X-Form** dialog box gives you improved access to frequently used options, and supports the following logical top-down workflow.

1. Select object.
2. Select poles to edit.
3. Change object degree.
4. Select movement method.
5. Edit poles.

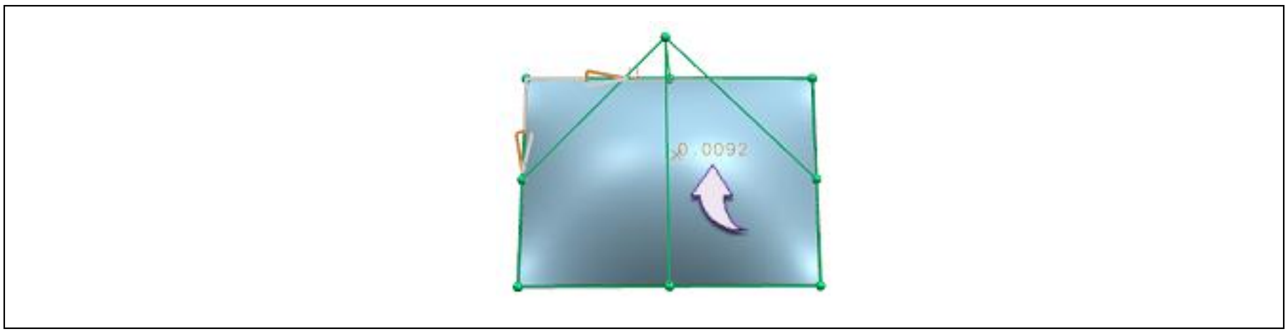
Other usability improvements include the following:

- You can now specify movement methods using options in the **Move, Rotate, Scale, Planarize** tabs.
- The **OrientXpress** tool is available to enable rapid selection of a movement method from within the scene.
- When you use **Move** with the **Polygon** option, polygon vector handles are displayed at each surface pole.



Vector handles on poles – dynamically controlled

- If you reduce the degree or the number of patches of a surface or curve, a temporary on-screen display shows the maximum deviation (at the point of instance).



On-screen maximum deviation

- A **Deselect Poles Automatically** check box has been added to the **Pole Selection** group:

  – Pole selection is cumulative; any pole selected is added to the previous selected pole. This is the default state.

  – When you select a pole, any other selected poles are automatically deselected.

- You can use the shortcut menu when you right-click a pole or polylines to:

  Increase or decrease the U and V degrees.

  Choose a movement method.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	(Modeling) **Freeform Shape → X-Form** (Shape Studio) **Edit → X-Form**
Menu	**Edit → Surface → X-Form**

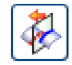
Match Edge

What is it?

The **Match Edge** command has been enhanced to include:

- The type of Match Edge operation is now calculated based on the selected **Reference** geometry so you don't need to explicitly select a type.
- Selection of both the **Edge to Edit** and a single **Reference** surface will occur automatically when the selection cursor crosses both the Edge to Edit and the Reference geometry.
- A **Shape Control** tab has been added:
  - You can control the amount of edge match applied to the matching surface using the **Modification Percentage** slider where 0% produces a totally unmatched state and 100% produces a fully matched state.
  - A **Blend** check box has been added to activate Blend, Depth and Skew input for matching operations.
- **Before** and **After** deviation values are displayed for checking edge matching based on your continuity specifications.
- You can use the **Align** parameterization option to enable the **Edge to Edit** surface to adopt to the parameterization of the **Reference** surface pole structure without increasing the degree or patches of the surface being matched.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	(Modeling) **Freeform Shape → Match Edge**  (Shape Studio) **Edit → Match Edge** 
Menu	**Edit → Surface →Match Edge**

Non-Proportional Zoom

What is it?

The **Non-Proportional Zoom** command has three main enhancements.

- You can dynamically designate the Non-Proportional Zoom aspect ratio of the display.
- You can set a Non-Proportional Zoom and change back and forth between the set Non-Proportional Zoom and the regular display.
- The **Non-Proportional Zoom Options** dialog box is where you can specify the behavior of your zoom operation.

The **Analyze Shape** toolbar has been updated to address the enhancements.

Display Non-Proportional Zoom

- Clicking this option will display the previously set Non-Proportional Zoom.
- Clicking this option more than once switches between the currently set Non-Proportional Zoom display and the regular display so you can go back and forth between displays.
- If no Non-Proportional Zoom display has been set, this option is unavailable.

Set Non-Proportional Zoom

- Clicking this option sets a Non-Proportional Zoom of the graphics display.
- If you click **Set Non-Proportional Zoom** when **Display Non-Proportional Zoom** is inactive, any previously set Non-Proportional Zoom will be lost.
- If a set Non-Proportional Zoom is displayed and you click **Set Non-Proportional Zoom**, the existing set Non-Proportional Zoom will be updated.

Non-Proportional Zoom Options

- Clicking this option opens the **Non-Proportional Zoom Options** dialog box.
- In the dialog box, you can specify the **Method** used for the Non-Proportional Zoom. There are two options:
  - **Rectangle** - The display aspect ratio is defined by dragging an explicit rectangle. This functions like previous releases.
  - **Dynamic** - The aspect ratio is dynamically defined by dragging the cursor. The non-proportional zoom is set once the left mouse button is released.
  - Using either method, you can now repeatedly adjust your Non-Proportional Zoom designation without leaving the function.
- In **Dynamic** mode, you can establish a center for your Non-Proportional Zoom using the **Anchor Center** check box:
  - **Anchor Center**
    - The Non-Proportional Zoom will always center on the first click of the cursor.
  - **Anchor Center**
    - The Non-Proportional Zoom will always center about the center of the view.
    - The screen will not pan upon release of the left mouse button.
- In **Dynamic** mode, you can use the **Sensitivity** slider to adjust the exaggeration of the Non-Proportional Zoom so larger or smaller zoom operations can be effected with one cursor drag.

Why should I use it?

- To evaluate the curvature flow of surfaces and curves when you edit poles using tools such as X-form or Studio Spline.
- To evaluate the curvature flow of surfaces and curves as a part of design quality validation such as checking the uniformity of the pole structure of curves, surfaces or sheet bodies.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	**Analyze Shape → Display Non-Proportional Zoom** **Analyze Shape → Set Non-Proportional Zoom** **Analyze Shape → Non-Proportional Zoom Options**

Menu	**View → Operation →Display Non-Proportional Zoom** **View → Operation → Set Non-Proportional Zoom** **View → Operation → Non-Proportional Zoom Options**

Edge Symmetry

What is it?

The new **Edge Symmetry** command matches edge geometry and continuity to a centerline plane for use in developing surfaces that are symmetrical across a plane.

- You can define the centerline plane by:
  - Selecting an existing plane.
  - Using one of the principal XYZ planes (XC-ZC is the default).
  - Using basic plane definition methods.
- You can specify an offset value from the designated plane using the **Offset** option.
- You can modify degree and patch parameterization.
- You can specify the method used for designating edge symmetry matching:
  - **Movement** – Use to specify if poles will be moved the minimum distance necessary to achieve the match (Normal), or if they will be moved using a projection algorithm to make the match (Project).
  - **Shape Control** – Use to control the position of selected poles and rows of poles and control the shape-blending factors of Blend, Depth and Skew.
- You can specify G0, G1, G2, and G3 continuity for edge symmetry matching and check before and after deviation values based on your specifications.

Where do I find it?

Application	Modeling and Shape Studio
Toolbar	(Modeling) **Freeform Shape → Edge Symmetry** (Shape Studio) **Edit → Edge Symmetry**
Menu	**Edit → Surface → Edge Symmetry**

Drafting

Chinese (GB) and Russian (ESKD) default standards settings

What is it?

Two new default standards files are available to configure specific annotation and drafting view preferences in accordance with Chinese and Russian drafting standards

Why should I use it?

Use either of these standards to automatically configure over 200 annotation and drawing view preferences for your PMI and Drafting environments.

Where do I find it?

To set the standard for the current NX session

Application	Drafting and PMI
Menu	**Tools→Drafting Standard**

To set the default standard or create a custom standard

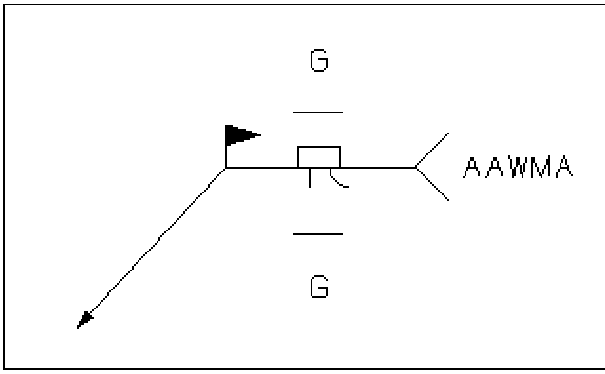
Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Drafting→General→Standard** tab

Weld symbol redesign and single flange JIS support

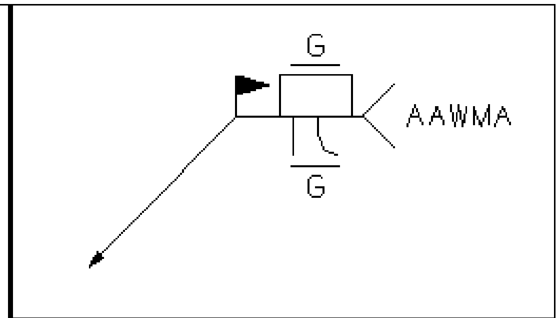
What is it?

The following enhancements are added to the Drafting and PMI weld symbols:

- The weld symbol is redesigned to display in a more standards-compliant manner. The display is based on the text height specified for the symbol.



Previous weld symbol



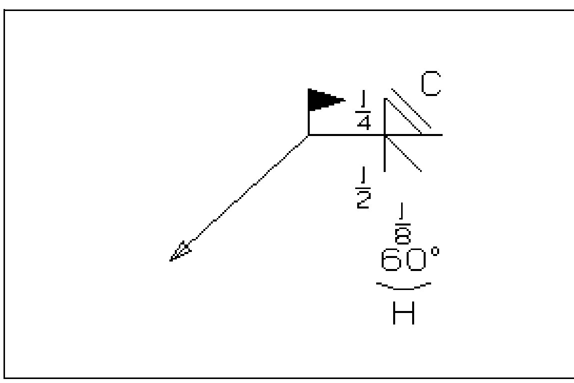
Current weld symbol

All new weld symbols will be created using this standards-compliant arrangement. Existing weld symbols will be updated to reflect the standards-compliant arrangement if the **Preferences→Drafting→General** tab→**Maintain Object Version On Update** option is not selected.

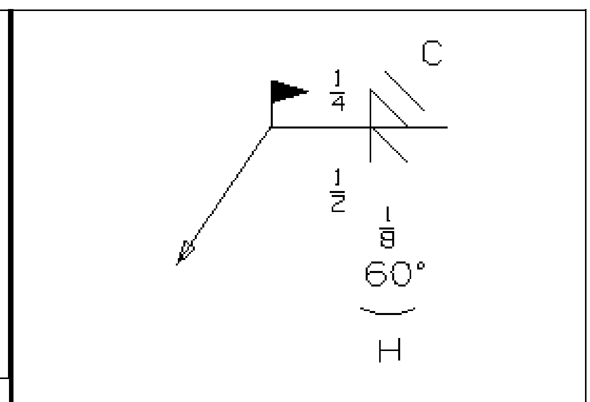
Caution

If you do not select the **Maintain Object On Update** option, all objects in the drawing will be updated, not just the weld symbols

- A new **Weld Space Factor** option provides additional control over the spacing between the different components of the weld symbol.

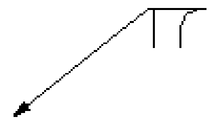


Weld Space Factor = 1



Weld Space Factor = 2

- Support for the creation of single flange JIS weld symbols is also provided.



JIS single flange weld symbol

Where do I find it?

Create or edit the **Weld Space Factor** of a weld symbol

Application	Drafting
Toolbar	**Annotation→Weld Symbol**
Menu	**Insert→Symbol→Weld Symbol**
Location in dialog box	**Settings** group

Set the preference and default of the **Weld Space Factor** for all weld symbols

Application	Drafting
Toolbar	**Drafting Preferences→Annotation Preferences**
Menu	**Preferences→Annotation** **File→Utilities→Customer Defaults**
Location in dialog box	**Drafting Preferences** dialog box →**Symbols** tab→**Weld** **Customer Defaults** dialog box →**Drafting→General→Standard** tab→**Customize Standard→Other Symbols→Weld** tab

NX Sheet Metal

 Resize Bend Radius enhancement

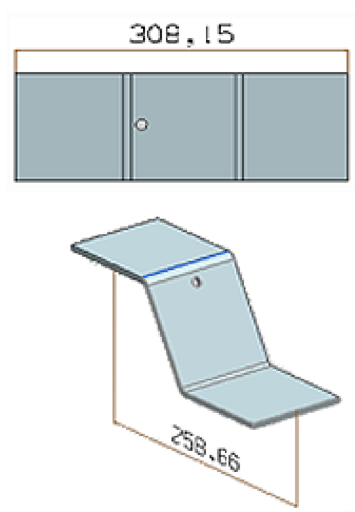
What is it?

A **Fixed Unfolded Length** type option is available. Use this option to resize selected bends while keeping a specified face or edge stationary.

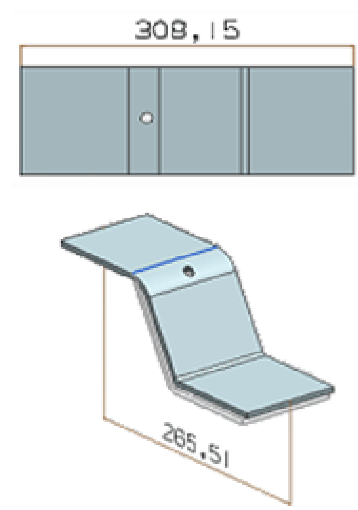
Select Face/Edge  is available to specify the stationary edge or face.

When you create the **Fixed Unfolded Length** type of **Resize Bend Radius** feature and change the bend radius, the dimensions of the unfolded model remain fixed, but the overall dimensions of the folded model change.

The figure on the right shows the Sheet Metal part after it was resized using the **Fixed Unfolded Length** type. The unfolded length of the Sheet Metal part remains the same, but the overall folded dimensions of the part are changed.



Initial bend radius = 3



Modified bend radius = 5

Why should I use it?

You can depict a bend in various intermediate stages, when you use the **Fixed Unfolded Length** type and resize selected bends in conjunction with the **Resize Bend Angle** command.

You can also:

- Study over-bending and springback effects of selected bends.
- Perform forming studies on an already-designed Sheet Metal part.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal→Resize Bend Radius**
Menu	**Insert→Resize→Resize Bend Radius**
Location in dialog box	**Type** group→**Fixed Unfolded Length** **Stationary Face/ Edge** group→**Select Face/Edge**

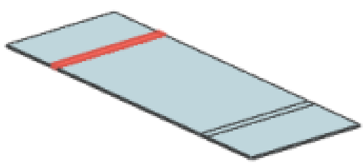
Rebend enhancement

What is it?

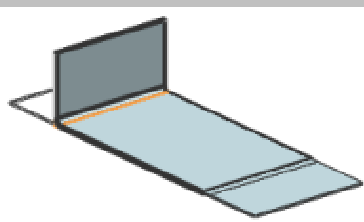
When you use the **Rebend** command, you can specify a face or an edge that remains stationary during the rebend operation.

You can control the positioning of the resultant body after the rebend operation.

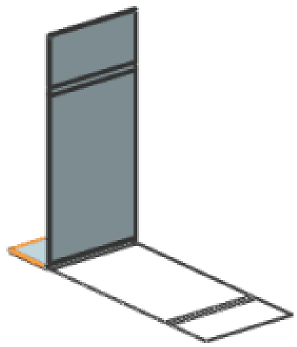
Select Face or Edge is available to specify the stationary face or edge.



Selected bend in the flattened Sheet Metal part for the rebend operation



Rebend generated without specifying a stationary face. The resulting body is positioned at its default location.



Rebend generated using a stationary face for reference. The positioning of the resulting body is changed.

Where do I find it?

Application	NX Sheet Metal, Flexible Printed Circuit Design
Toolbar	**NX Sheet Metal→Rebend** **Flexible Printed Circuit Design→Rebend**
Menu	**Insert→Form→Rebend**
Location in dialog box	**Stationary Face or Edge** group→**Select Face or Edge**

Chapter 6: Manufacturing

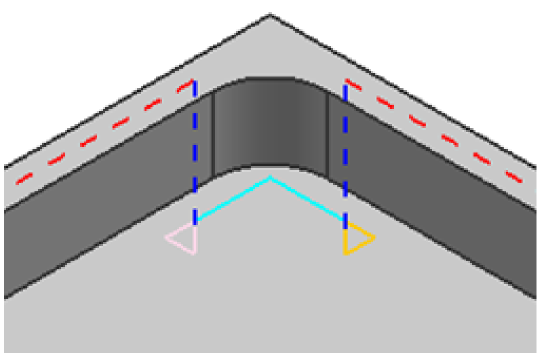
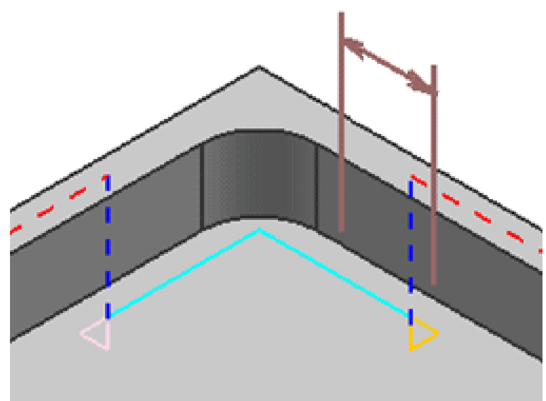
Planar Milling

Overlap Distance

What is it?

The **Overlap Distance** option for the **Planar Mill**, **Planar Profile**, **Profile 3D** and **Solid Profile 3D** operation subtypes has changed. It now affects the current operation rather than the following clean up operation. The location of the option has also changed.

During a sequence of rest milling operations, use the **Overlap Distance** option to extend the path for the current operation so that it overlaps the cut region of another operation. **Overlap Distance** is only available when you use the **Use 2D IPW** or **Use Reference Tool** options.

 **Overlap Distance** set to zero	 **Overlap Distance** set to a value of 15.00

Why should I use it?

Using this option helps you eliminate scallops and obtain a full clean up between tool paths.

Where do I find it?

Application	Manufacturing
Prerequisite	For **Planar Mill**, **Planar Profile**, **Profile 3D** and **Solid Profile 3D** operation subtypes, the **In Process Workpiece** option must be set

	to **Use 2D IPW** or **Use Reference Tool**.
Menu	**Insert→Operation**
Location in dialog box	[Planar Milling] operation dialog box→**Path Settings** group→**Cutting Parameters**  →**Containment** tab→**Overlap** group

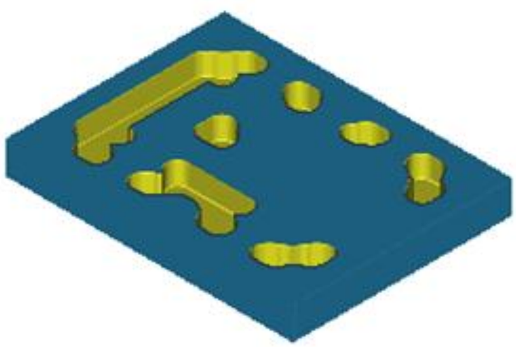
Reference Tool

What is it?

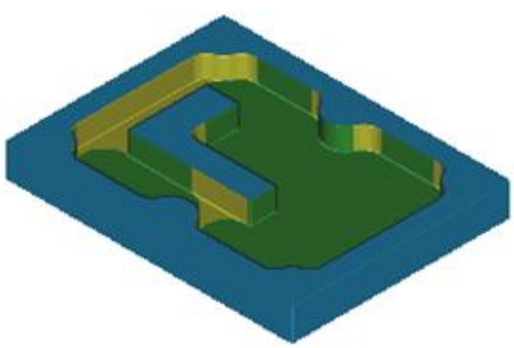
The **Reference Tool** option is now available in the **Planar Mill**, **Planar Profile**, **Profile 3D** and **Solid Profile 3D** operation subtypes.

Use the **Reference Tool** option to create a new operation with a smaller tool that references a larger tool. The smaller tool only removes material that would not have been cut by the larger tool. You can place the reference tool operation before or after operations with larger tools.

Place the reference tool operation before operations using the larger tool to first rough cut corners with a smaller tool.

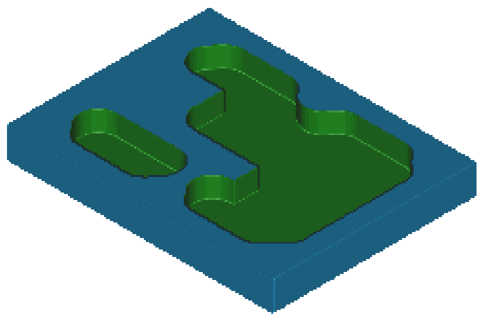


The yellow area shows the areas cut with the smaller tool. The tool path was contained in those areas by the reference tool.

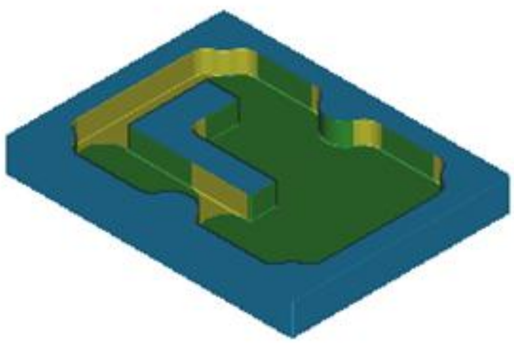


The green area shows the areas cut with the larger tool. The tool path was previously contained in the yellow areas by the reference tool.

Place the reference tool operation after operations using the larger tool to cut areas that the larger tool could not fit into.



The part has been roughed with a large tool. The reference tool operation will remove the remaining material.



The yellow area shows the areas cut with the smaller tool. The tool path was contained in the yellow areas by the reference tool.

Why should I use it?

Using this option helps you eliminate tool motion where there is no material. The tool path can be created before or after the roughing paths.

Where do I find it?

Application	Manufacturing
Prerequisite	For **Planar Mill**, **Planar Profile**, **Profile 3D** and **Solid Profile 3D** operation subtypes, the **In Process Workpiece** option must be set to **Use Reference Tool**.
Menu	**Insert→Operation**
Location in dialog box	[Planar Milling] operation dialog box→**Path Settings** group→**Cutting Parameters** →**Containment** tab→**Reference Tool** group

Use 2D IPW

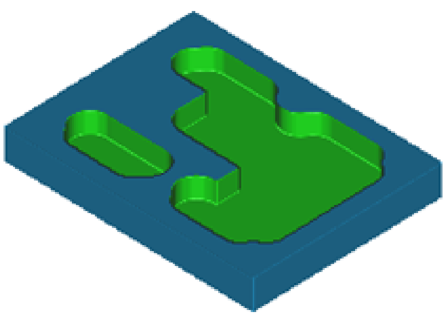
What is it?

The **Use 2D IPW** option replaces the **Save 2D In Process Workpiece** option.

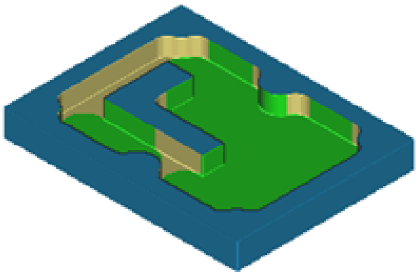
Use the **Use 2D IPW** option in Planar Milling to machine only the material that is left on the part in subsequent operations. The software tracks the remaining material from one operation to the next.

When you use this option, the software:

- Finds uncut material by evaluating previous Planar Milling operations that share the same part geometry.
- Only recognizes valid Planar Milling operations including **Planar Profile**, **Profile 3D**, **Solid Profile 3D**, and **Planar Mill**. If other operations are found, the software ignores them and uses the 2D IPW from the last Planar Milling operation.
- Supports the use of ball endmills.



The first tool path roughs out most of the material.



The second tool path removes only the remainder of the material recognized in the IPW.

Why should I use it?

When you use this option, you avoid tool motion where material has been removed by previous operations.

Where do I find it?

Application	Manufacturing
Prerequisite	You must be in a **Planar Mill**, **Planar Profile**, **Profile 3D** or **Solid Profile 3D** operation.
Menu	**Insert**→**Operation**
Location in dialog box	[Planar Milling Operation] dialog box→**Path Settings** group→**Cutting Parameters** →**Containment** tab→**Blank** group→**In Process Workpiece** list→**Use 2D IPW**

Output Contact/Tracking Data

What is it?

The **Output Contact/Tracking Data** option is now available for **Profile 3D** and **Solid Profile 3D** operations.

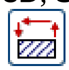
Use the **Output Contact/Tracking Data** option to set Contact Contour cutter compensation. When you use this option, the cutter compensation is output for several tool contact locations rather than one tool end location, for all cutting motions in an NC operation.

Contact Contour is also referred to as full radius cutter compensation or material edge contouring. Use contact contour cutter compensation for tool paths generated at the circumference of the tool radius.

Why should I use it?

Use this option when the machine operator will enter the full tool size, instead of the variation from the programmed size.

Where do I find it?

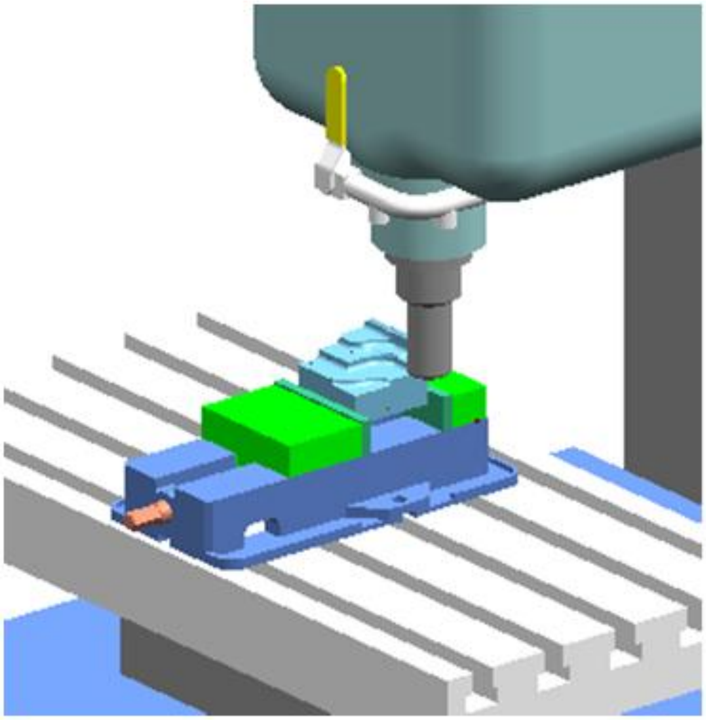
Application	Manufacturing
Prerequisite	The operation must have a finish pass or profile pass.
Menu	**Insert→Operation**
Location in dialog box	**Profile 3D**, **Solid Profile 3D** operation dialog box→**Non Cutting Moves**  →**More** tab→**Cutter Compensation** group

ISV

Automatic Part, Blank, and Check geometry in ISV

What is it?

ISV recognizes the **Part**, **Blank** and **Check Geometry** defined for an operation. It is not necessary to define the geometry again in the **Machine Tool Navigator**.



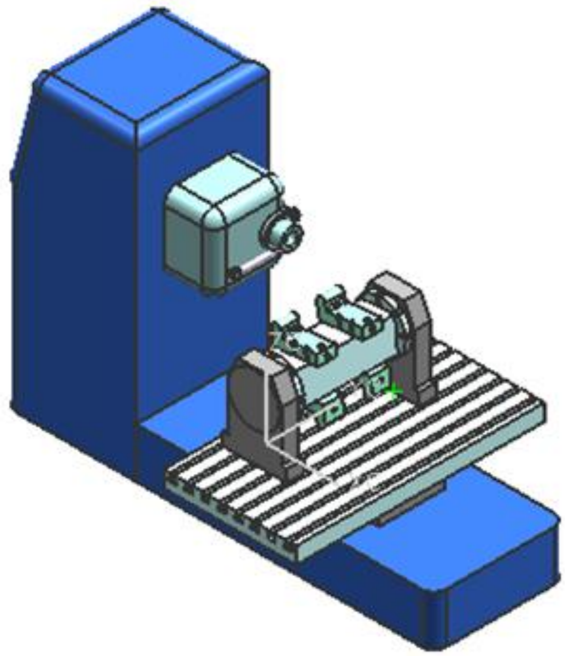
Where do I find it?

Application	Manufacturing
Prerequisite	Operations with defined geometry and generated tool paths. A defined machine assembly.

Preview Motion

What is it?

Use the **Preview Motion** command to control the position of each machine axis and view it relative to the other axes.



Why should I use it?

You can view the machine tool motion of all axes from one dialog box without simulating the tool path. You can also dynamically move each axis between its limits, as you would move an axis on the real machine tool.

Where do I find it?

Application	Manufacturing or Machine Tool Builder
Prerequisite	A defined machine assembly.
Machine Tool Navigator	Right-click in the background→**Preview Motion**

Synchronization

What is it?

You can now view synchronization status in the **Channel** column of the **Operation Navigator**. You can find program groups with synchronization data, and delete the synchronization, directly from the **Operation Navigator**.

Why should I use it?

This allows you to remove synchronization without opening the Synchronization Manager.

Where do I find it?

Application	Manufacturing
Prerequisite	A Program parent group with synchronization data
Operation Navigator	Right-click the Program group→**Toolpath**→**Synchronize** Right-click the Program group→**Toolpath**→**Remove Synchronization**

Post Builder

Post Builder support for Siemens Sinumerik controllers

What is it?

The following template posts support Siemens Sinumerik controllers:

- Sinumerik_840D supports milling machine with the Sinumerik 840D controller.

  Support includes **CYCLE800** and **TRANS/AROT** coordinate rotation output. To output coordinate rotation codes, you must set a proper LOCAL MCS with CSYS Rotation special output.

- Sinumerik_840D_lathe supports 2-axis turning machine with the Sinumerik 840D controller.
- Sinumerik_828D supports milling machine with the Sinumerik 828D controller.
- Sinumerik_802D_3axis supports 3-axis milling machine with the Sinumerik 802D controller.

You can add the following user-defined start events to support the Siemens controllers:

- Add a start event for the Sinumerik 840D controller to a single milling operation, or to the lowest level program group in the Program view of the Operation Navigator. The default parameter settings in the Sinumerik 840D start event dialog box are the same as the default settings in the Sinumerik 840D template post.

  If you choose *Swiveling* in the **Transformation** option of the Sinumerik_840D user-defined event, CYCLE800 is output and transformation is turned off (**TRAFOFF**), otherwise **TRANS/AROT** is output for coordinate rotation.

  For drilling cycles, if the work plane is not the XY plane, coordinate rotation codes could be output without the **LOCAL MCS** setting.

  The **Sinumerik 840D** user-defined event is available in the Sinumerik_840D template post.

- Add the **DNC Header** user-defined event to the top working program group under NC_PROGRAM so that the NC file can be accessed by the Siemens MCIS (Motion Control Information System).

  The **DNC Header** user-defined event is available in all milling template posts.

- Add the **Sinumerik Program Control** user-defined event to a program group so that you can call external subroutines.

The **Sinumerik Program Control** user-defined event is available in the Sinumerik_840D and Sinumerik_828D template post.

Note

DNC Header and **Sinumerik Program Control** user-defined events should not be assigned to an operation.

You can set the Sinumerik version in Post Builder with the custom command PB_CMD_Sinumerik_Version in Sinumerik_840D postprocessors.

Sinumerik version 6 and later support CYCLE832 high speed machining G code output.

Where do I find it?

Post Builder template posts

Application	Post Builder
Menu	**File→New**
Location in dialog box	**Controller** group→**Library** list

Custom commands

Application	Post Builder
Location in dialog box	**Program & Tool Path** tab→**Custom Command** tab

Sinumerik 840D user-defined events

Application	Manufacturing
Menu	**Insert→Operation**
Location in dialog box	Operation dialog box→**Machine Control** group→**Edit**  (**Start of Path Events**)

Improved Post performance

What is it?

You can add the new custom command **PB_CMD_activate_Fanuc_turbo_mode** to the Start of Program event of the Program Start Sequence. This command improves the postprocessing performance of Fanuc style posts for four and five-axis machines.

There are also general improvements to NX Post performance.

Where do I find it?

Import the **PB_CMD_activate_Fanuc_turbo_mode** custom command.

Application	Post Builder
Location in dialog box	**Program & Tool Path** tab→**Custom Command** tab→**Import**
File location	*POSTBUILD/pblib/custom_command/pb_cmd_activate_turbo_mode.tcl*

Add the custom command to the Start of Program event.

Application	Post Builder
Location in dialog box	**Program & Tool Path** tab→**Program** tab→**Program Start Sequence** marker

Parallel Generate

What is it?

Use the **Parallel Generate** command to generate operations in the background while you continue working. Select any number of operations at the same time, and they count as one parallel process. You can:

- Run multiple processes concurrently.
- Stop parallel processes with the **Stop Parallel Generate** command.

Note

- You must select all the operations that use an In-Process Workpiece together and submit them as one process.
- *The part must remain open in the current session.* You can continue to work on it, or work on another part, but you cannot close the part. This is different than batch processing, which requires you to close the part and wait until the batch processing is complete to open the part.

Use the **Maximum Concurrent Processes** customer default to specify how many processes the software can run concurrently. The maximum number is four.

New operation status indicators include the following:

- **Pending**

  This indicates that the operation is in queue for a scheduled process. The indicator appears when the maximum number of concurrent processes is exceeded.

- **Parallel Generating**

  This indicates that the operation is processing in the background.

Tip

When the background processing is complete, the operation status changes to **Repost**. There is no notification message when the process is complete.

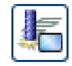
Why should I use it?

Using the **Parallel Generate** command helps if:

- Your computer has multiple cores or processors, and you want to use them to speed up tool path processing.
- Your computer has a slower processor, and you do not want to wait for complex tool paths to generate before working on another operation.

Where do I find it?

Parallel Generate commands

Application	Manufacturing
Toolbar	**Operations→Parallel Generate Tool Path** 
Operation Navigator	Right-click the selected operations→**Parallel Generate**/**Stop Parallel Generate**

Maximum Concurrent Processes customer default

Application	Manufacturing
Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Manufacturing→Operation→General** tab→**Parallel Tool Path Generation** group

Store postprocessor time in Teamcenter

What is it?

Machining time calculated by the postprocessor is automatically stored in the Duration property of an Activity in Teamcenter.

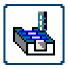
Where do I find it?

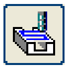
Application	Manufacturing
Prerequisite	Operations with tool path generated.
Operation Navigator	Right-click the Program group→**Tool Path**→**Synchronize** Right-click the operation or Program group→**Post Process**

Face Milling-Negative Floor Stock

What is it?

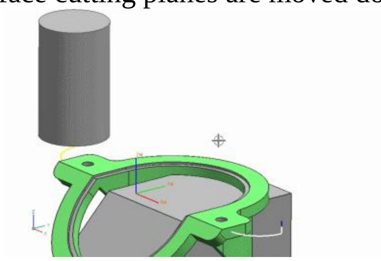
You can now specify a negative **Final Floor Stock** value for the following Face Milling operation subtypes:

 **Face Milling Area**

 **Face Milling**

 **Face Milling Manual**

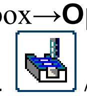
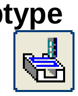
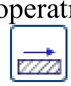
All moves created on the face cutting planes are moved down by the specified value.



Why should I use it?

Entering a negative **Final Floor Stock** value is useful when you have to make a minor adjustment for a modeled dimension to meet a functional requirement. For example, when you have to shorten the height of a face or move a face into the part.

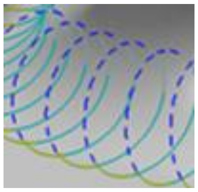
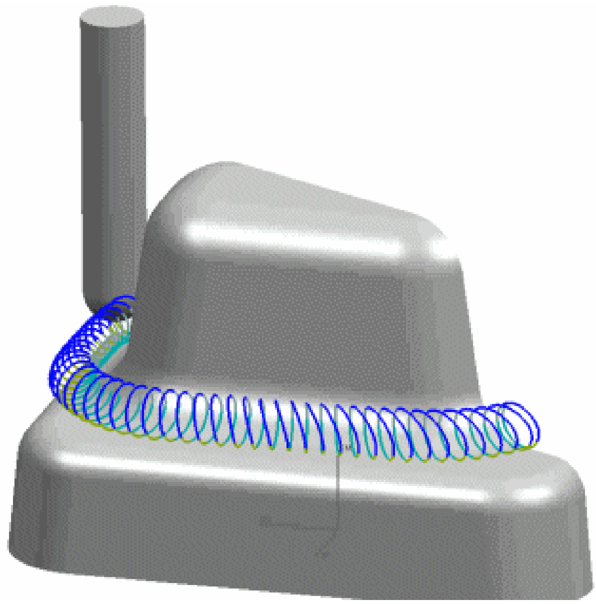
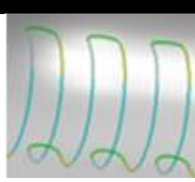
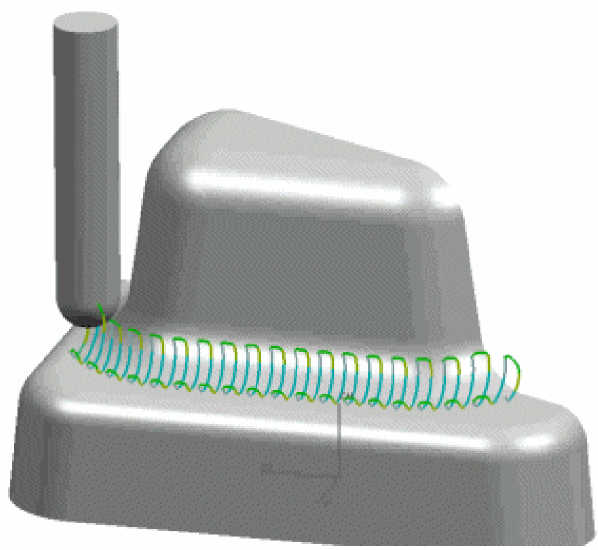
Where do I find it?

Application	**Manufacturing**
Menu	**Insert**→**Operation**
Location in dialog box	**Create Operation** dialog box→**Operation Subtype** group→**Face Milling Area**  /**Face Milling**  /**Face Milling Manual**  Face Milling operation dialog box→**Path Settings** group→**Cutting Parameters**  → **Stock** tab→**Stock** group→**Final Floor Stock**

Non Cutting Moves Enhancement Smooth Traverse

What is it?

A **Traverse Type**, **Smooth** option is added to non cutting moves in fixed and variable contouring operations.

 Traverse Feedrate	
 Stepover Feedrate	

Why should I use it?

This traverse type lets the tool merge tangentially from a cut, retract, or approach move, to the next cut, engage, or approach move without slowing down.

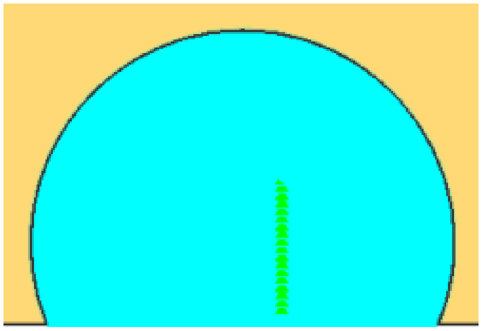
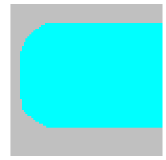
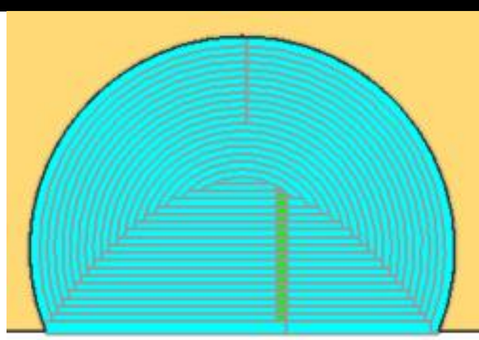
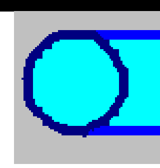
Where do I find it?

Application	**Manufacturing**
Dialog Box	[Surface Contouring Milling] operation→**Non Cutting Moves**
Location in dialog box	**Transfer/Rapid** tab→**Between Regions and Within Regions** group→**Traverse Type**→**Smooth**

Fill and Silhouette Fill display options

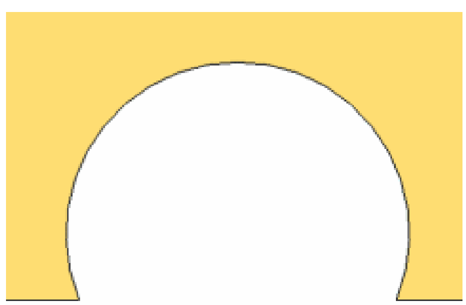
What is it?

Fill and **Silhouette Fill** display options are added to help you view the area cut by the wire.

	 Fill
	 Silhouette Fill

Why should I use it?

It can be difficult to see the area being cut when displaying the tool path in a wireframe silhouette. These display options shade the actual cut operation, so that you can ensure that all the material within a region or boundary is cut.



Where do I find it?

Application	Manufacturing
Location in dialog box	Operation dialog box→Tool Path group→Edit Display→Path Display

Preview projects

What are preview projects?

What is it?

Preview projects are intended for production release in the future.

Why should I use it?

The preview gives you an opportunity to do some real world testing, and then give us some feedback before the products are officially released.

Where do I find it?

To test preview projects, contact your customer support representative.

Inspection Programming

What is it?

Inspection Programming enables you to quickly and accurately define complete, verifiable machine inspection programs for Coordinate Measuring Machines (CMMs). Based on the NX CAM architecture, Inspection Programming uses:

- Part geometry from the NX solid model.
- Automatic generation of features and tolerance information from NX Product Manufacturing Information (PMI).
- Full machine simulation using NX collision detection.

Why should I use it?

Because Inspection Programming is fully integrated with NX and PMI, annotations are easily available on the CAD model and can encapsulate requirements as well as part information. CMM inspection programs, which are stored and maintained in Teamcenter, can be quickly constructed without any required programming knowledge. The Inspection Programming preview showcases:

- Functionality for creating feature, tolerance, and paths manually.
- Three-axis scanning.

- Generic Dimensional Measurement Interface Standard (DMIS) output.

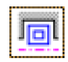
Inspection Programming aims to eliminate expensive manual inspection gages, reduce tooling costs and machine reprogramming time, and better analyze features and tolerances against measured data.

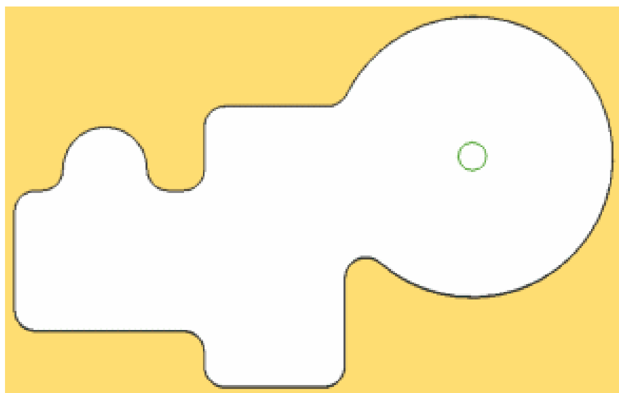
Where do I find it?

To test Inspection Programming, contact your customer support representative.

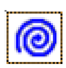
Wire/EDM No-core Follow Periphery

What is it?

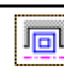
You can generate a continuous cut using the **Follow Periphery**  cut pattern option. You can either create a thread hole, or have the system infer it. This cut pattern cuts from the inside out to ensure that all material with the boundaries is removed.



Why should I use it?

Use this option when the **Spiral**  cut pattern cannot remove all material within the boundaries, or give you the desired surface finish.

Where do I find it?

Application	**Manufacturing**
Prerequisite	Type must be set to **wire_edm** in the **Create Operation** dialog box. The **NOCORE** operation subtype must be selected.
Location in dialog box	**NOCORE→Cut Pattern→Follow Periphery** 

Chapter 7: Systems Design

PCB Exchange

PCB.xchange application name change

What is it?

The PCB.xchange application is now called PCB Exchange.

Space Systems Thermal solution

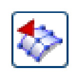
What is it?

You can now create a Space Systems Thermal solution from your PCB Exchange model by selecting the **Space Thermal** option in the **Create ESC Solution** dialog box.

Why should I use it?

Use the **Space Thermal** option when you want to solve a thermal model of your PCB using the **Space Systems Thermal** simulation.

Where do I find it?

Application	PCB Exchange
Menu	**PCB Exchange** → **Thermal/Flow Simulation** → **Create ESC Solution**
Toolbar	**PCB Exchange** → **Create ESC Solution** 
Location in dialog box	**Output Option** group → **Write Boundary Conditions** list → **Space Thermal**

View or modify a specific NX object

What is it?

When you select an NX object, the new **Object** and **Feature** commands bring up the dialog box that shows the right attributes applied to this object. For example, PCB Exchange brings up the **Keep-in Area Attributes** dialog box if the specified NX object was designed as a keep-in area.

- The **Object** command lets you select an NX object from the graphics window.

- The **Feature** command lets you select an NX object from a list in the **Feature Selection** dialog box listing all features in the model.

If the selected NX object has no PCB attributes, PCB Exchange issues an error message.

If you select more than one object, PCB Exchange shows the PCB attributes of the first selected object.

Why should I use it?

These new commands help you to quickly find what PCB Exchange attributes apply to a specific NX object. It is very useful when you have many objects in the model and you don't know what they are.

Where do I find it?

Application	PCB Exchange
Menu	**PCB Exchange** → **Edit Attributes** → **Object** **PCB Exchange** → **Edit Attributes** → **Feature**

PCB Exchange for Zuken

PCB.xchange + Zuken application name change

What is it?

The PCB.xchange + Zuken application is now called PCB Exchange for Zuken.

Chapter 8: Digital Simulation

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.

NX7

Solver	File Type	NX 7
NX Nastran	Import ASCII (.dat)	6.1
	Import Binary (.op2)	6.1
	Export ASCII (.dat)	6.1
	Post-processing of Results	6.1
MSC Nastran	Import ASCII (.dat)	2008r1
	Import Binary (.op2)	2008r1
	Export ASCII (.dat)	2008r1

Solver	File Type	NX 7
	Post-processing of Results	2008r1
Abaqus	Import ASCII (.inp)	6.8-1
	Import Binary	N/A
	Export ASCII (.inp)	6.8-1
	Post-processing of Results	6.8-EF2
ANSYS	Import ASCII (PREP7, CDB)	12
	Import Binary (.rst, .rth)	12
	Export ASCII (.inp)	12
	Post-processing of Results	12
LS-DYNA	Import ASCII	N/A
	Import Binary	N/A
	Export ASCII (.k)	971R3.2.1
	Post-processing of Results	N/A

NX 6 releases

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3
NX Nastran	Import ASCII (.dat)	6.0	6.1	6.1	6.1
	Import Binary (.op2)	6.0	6.1	6.1	6.1
	Export ASCII (.dat)	6.0	6.1	6.1	6.1
	Post-processing of Results	6.0	6.0	6.1	6.1
MSC Nastran	Import ASCII (.dat)	2007r1	2008r1	2008r1	2008r1
	Import Binary (.op2)	2007r1	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2007r1	2008r1	2008r1	2008r1
	Post-processing of Results	2007r1	2008r1	2008r1	2008r1
Abaqus	Import ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1
	Import Binary	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1
	Post-processing of Results	6.7-5	6.8-1	6.8-3	6.8-EF2

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3
ANSYS	Import ASCII (PREP7, CDB)	11	11 SP1	11 SP1	11 SP1
	Import Binary (.rst, .rth)	11	11 SP1	11 SP1	11 SP1
	Export ASCII (.inp)	11	11 SP1	11 SP1	11 SP1
	Post-processing of Results	11 SP1	11 SP1	11 SP1	11 SP1
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R2	971R2	971R3.2.1	971R3.2.1
	Post-processing of Results	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

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
	Post-processing of Results	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.8-1
ANSYS	Import ASCII (PREP7, CDB)	10	10	11	11	11	11	11
	Import Binary (.rst, .rth)	10	10	11	11	11	11	11
	Export ASCII (.inp)	10	10	11	11	11	11	11
	Post-processing of Results	10	11	11	11	11	11	11 SP1

NX 4 releases

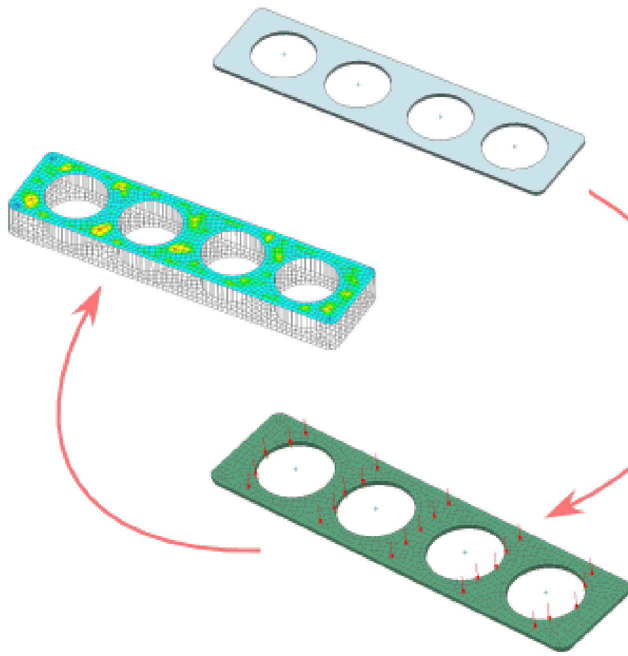
Solver	File Type	NX 4	NX 4.0.1	NX 4.0.2	NX 4.0.3	NX 4.0.4
NX Nastran	Import ASCII (.dat)	4.0	4.1	4.1	5.0	5.0
	Import Binary (.op2)	4.0	4.1	4.1	4.1	4.1
	Export ASCII (.dat)	4.0	4.1	4.1	5.0	5.0
	Post-processing of Results	4.0	4.1	4.1	5.0	5.0
MSC Nastran	Import ASCII (.dat)	2005	2005	2005	2005	2005
	Import Binary (.op2)	2005	2005	2005	2005	2005
	Export ASCII (.dat)	2005	2005	2005	2005	2005
	Post-processing of Results	2005	2005	2005	2005	2005
Abaqus	Import ASCII (.inp)	6.5-1	6.5-1	6.5-1	6.6	6.6
	Import Binary	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.5-1	6.5-1	6.5-1	6.6	6.6
	Post-processing of Results	6.5-1	6.5-1	6.5-1	6.6	6.6-3
ANSYS	Import ASCII (PREP7, CDB)	8	9	9	10	10
	Import Binary (.rst, .rth)	8	9	9	10	10
	Export ASCII (.inp)	8	9	9	10	10

Solver	File Type	NX 4	NX 4.0.1	NX 4.0.2	NX 4.0.3	NX 4.0.4
	Post-processing of Results	9	9	9	10	10

Gasket analysis support

What is it?

For NX Nastran, ANSYS, and Abaqus models, this release includes enhancements that make it possible for you to model and analyze gaskets. Gaskets are relatively thin components which you can place between two bodies to create a sealing effect and prevent the leakage of fluid. Gaskets can be made of many different materials, such as rubber or composites. The gasket material itself is usually under compression.

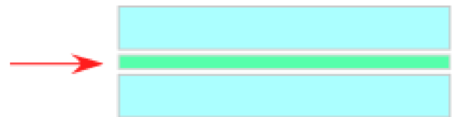


To support gasket modeling, this release includes:

- For NX Nastran or ANSYS, a new **Gasket** material type option in the **Assign Material** dialog box. When you select **Gasket** as the material type, you can then use the new **Gasket Material** dialog box to define all the necessary properties for modeling a gasket material.
- A new Gasket Results page in the **Structural Output Requests** dialog box to request the output of gasket results for NX Nastran analyses. See **Unsatisfied xref title** for more information.
- For Abaqus, support for several different types of Abaqus gasket elements, as well as support for a new **Gasket Section** type of physical property table and **Gasket Behavior** material. See **Unsatisfied xref title** for a complete list of the supported elements.

General gasket modeling process

Although the details of the process for modeling and analyzing gaskets differ depending on your specified solver, the overall process fairly standard. In general, you model a gasket as a very thin solid body that lies between two other solid bodies.



You mesh the gasket itself, along with the bodies on either side, with solid elements. Generally, the mesh on the gasket is only a single layer of solid elements. To ensure the proper connectivity in a gasket analysis, you must ensure that the nodes from the gasket mesh are connected to the nodes in the meshes on the surrounding bodies. The specific requirements for how you establish this connection depend on your solver. In general, you can either:

- Ensure that the elements from the gasket mesh share nodes with the elements on the two adjacent bodies. In Advanced Simulation, you can use a **Glue Coincident** type of **Mesh Mating Condition**, for example, to ensure that the nodes are shared. You can also merge nodes between the adjacent bodies.
- Connect the nodes on the gasket mesh to the nodes on the surrounding bodies with constraints, such as an **MPC** type of **Manual Coupling**.

General capabilities

Command Finder for CAE

What is it?

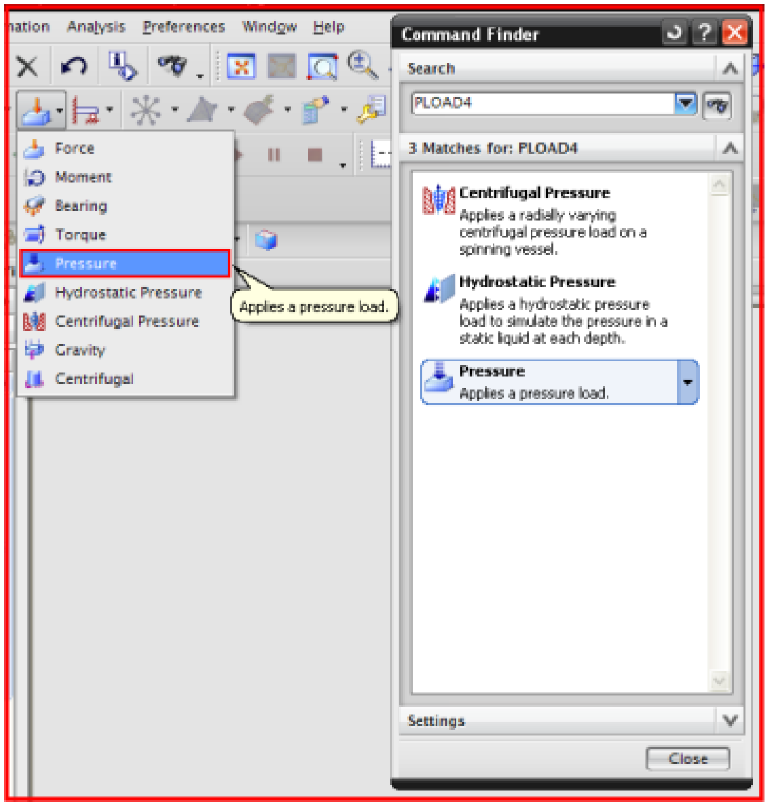
Command Finder is a search tool that helps you find a specific NX command associated with the words or phrases you enter.

For this release, **Command Finder** now supports all commands in the Advanced Simulation and Design Simulation applications.

From the list of commands, you can:

- Display the command location.
- Launch the command, if it is available in the current application.
- Display the Help information for the command.

For boundary conditions, you can also search on the solver input name, such as **PLOAD4** for an NX Nastran **Pressure** load.



Use customer defaults for the Command Finder to:

- Save a cached file of all menu and toolbar commands.
- Identify a location for a custom list of words and phrases associated with specific commands.
- Search for a command using a secondary language.

Where do I find it?

Command Finder customer defaults

Menu	**File→Utilities→Customer Defaults→Gateway→User Interface**
Location in dialog box	**Command Finder** tab

Command Finder command

Application	Advanced Simulation or Design Simulation
Menu	**Help→Command Finder**
Toolbar	Standard toolbar→**Command Finder** 

Display controls

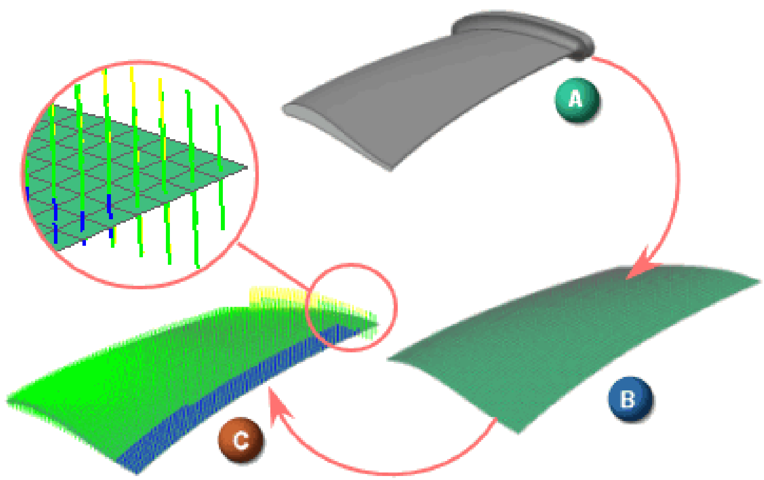
Displaying 2D element thickness values

What is it?

You can use the new **Thickness Information** command in the **Simulation Navigator** to display the 2D element thickness values in your model. This lets you verify the thickness values the software will apply when you either export or solve your model. Advanced Simulation provides you with a variety of ways to define the thickness of 2D elements. For example, the thickness values can be derived from the thickness of a midsurface or specifically assigned through either **Element Associated Data** or a physical property table. Because of the multitude of methods for specifying element thickness, you need a way to verify that those thickness values are defined correctly prior to solving your model. You can also use **Thickness Information** to create a spatial field of the 2D element thickness values.

Creating a display of element thickness values

Thickness Information creates a color-coded line display that shows the general statistical distribution of the thickness values across your 2D mesh. The following example shows a turbine blade model (A) which we used to create and mesh a midsurface (B). (C) shows the thickness display for the 2D elements on that midsurface.



- The lines represent the actual thickness values of the elements. The software draws those lines along each element's normal at the element's center of gravity. You can use the **Line Scale** option in the **Thickness Information** dialog box to control the overall length of the lines.
- The colors represent the range of thickness values across the model. These colors let you visually verify that the thickness values are properly distributed across the mesh. You can use options in the **Thickness Information** dialog box to control the colors the software assigns to each thickness range.
  - **Mid Range Thickness** elements have a thickness that is within one standard deviation (σ) either above or below the mean thickness value for the mesh.
  - **Max Range Thickness** elements have a thickness that is greater than one standard deviation above the mean thickness value for the mesh.

- **Min Range Thickness** elements have a thickness that is less than one standard deviation below the mean thickness value for the mesh.
- **Zero Thickness** elements do not have an assigned thickness value.

You can use a thickness display to quickly identify:

- Any sudden changes in color that may indicate incorrectly assigned thickness values.
- Elements that do not have an assigned thickness.

Creating fields from element thickness displays

You can also use the **Thickness Information** command to create a spatial field of thickness values from the 2D elements in your model. In the **Thickness Information** dialog box, when you select the **Create Field** option, the software generates a new field and places it in the **Fields** node in the **Simulation Navigator**.

Creating a thickness field can be helpful if you are working with a model in which the thickness of mesh is being derived from a midsurface. On a large model, the derivation of thickness data from a midsurface can be time and resource intensive since the software must recalculate the thickness each time you use **Thickness Information** to generate a display. If you are iterating through the meshing process with different element sizes, you can use the new **Create Field** option to generate a field from the midsurface data the first time you use the **Thickness Information** command. You can edit the **Mesh Associated Data** for that mesh and change the specified **Thickness Source** from **Midsurface** to **Field**. You can then select the newly created thickness field.

The ability to create a thickness field from a display of your 2D element thicknesses also allows you to edit or copy that thickness data. You can then reuse that modified data in either the same model or another model. For example, you can import a Nastran bulk data file that contains a coarse mesh in which the element thickness values are defined directly on the CQUAD4 elements. You can then use the new **Create Thickness Field** capability to generate a thickness field from that data. If you then remesh the model with a finer mesh, you can use that thickness field to map the original thickness values to the new mesh

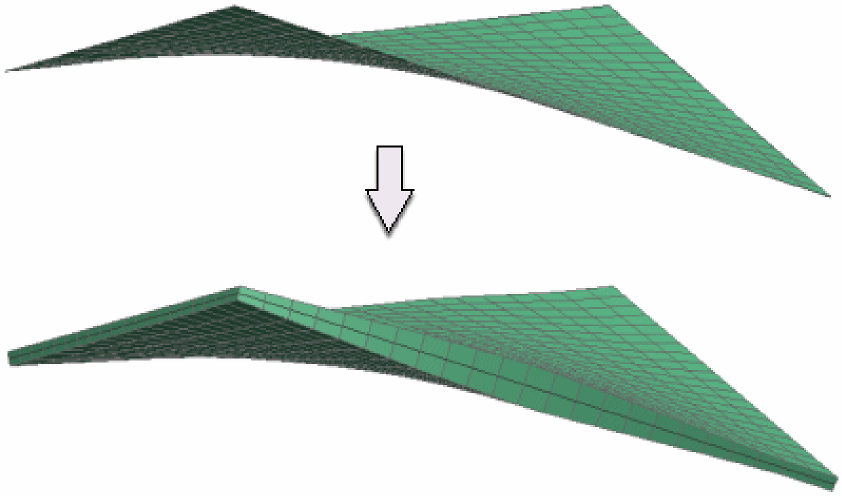
Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains a 2D mesh
Simulation Navigator	Right-click the appropriate 2D mesh collector or 2D mesh and choose **Thickness Information**

Persistently displaying 2D element thickness

What is it?

Use the **Display 2D Element Thickness** option in the **Mesh Display** dialog box to display a graphical representation of the thickness of the 2D elements in your model. When you select this option, the software displays the 2D elements as if they were solid elements. The software continues to display the thickness of the elements until you turn this option off.



With the **Display 2D Element Thickness** option, the software displays a thickness for all 2D elements in the model, regardless of how the thickness is defined. For example, the software displays element thicknesses that are:

- Derived from a midsurface's thickness
- Defined in a physical property table
- Defined as **Element Associated Data** for selected elements
- Defined as a field.

Why should I use it?

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	An active FEM file that contains 2D elements with defined thickness values.
Menu	**Preferences→Mesh Display**
Simulation Navigator	In the FEM, right-click the 2D mesh collector or 2D mesh and choose **Edit Display**. Select **Display 2D Element Thickness**. In the Simulation file, right-click the 2D mesh collector or 2D mesh and choose **Create Display Override** or **Edit Temporary Display**. Select **Display 2D Element Thickness**.

Boundary conditions

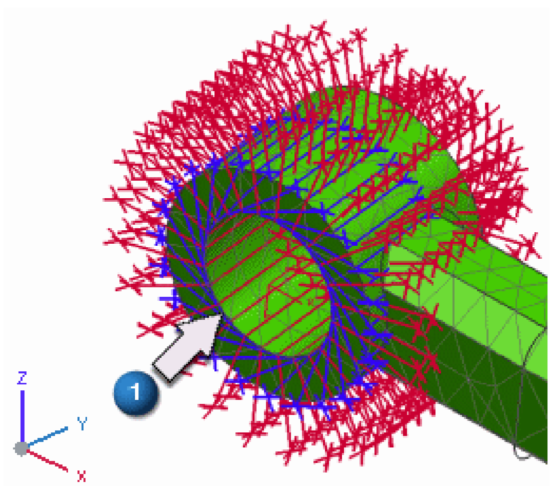
Improvements to constraint resolution

What is it?

Improvements have been made to the method in which the software handles two constraints defined on shared geometry or nodes. Also, for Simulation files created in Advanced Simulation, the constraint resolution workflow has been changed so that you are now in control of initiating the resolution.

Constraints on shared geometry or nodes resolve correctly

In previous releases, two constraints defined on shared geometry or nodes, regardless of the constrained degrees of freedom, were considered to be in conflict. However, in some cases, the degrees of freedom affected by two intersecting constraints are not actually in conflict, as shown in the example below.



Edge indicated by (1) is shared by both constraints

User Defined Constraint (blue, defined on outer face)	Pinned constraint (red, defined on inner face)
DOF1=Free DOF2=Fixed DOF3=Free DOF4=Free DOF5=Free DOF6=Free	A pinned constraint uses a cylindrical coordinate system. R and Z directions are fixed Theta (rotating) direction is free All rotational DOF are also free In this example, the Cartesian coordinate system for the User Defined constraint is oriented such that the Y direction is parallel with the Z direction in the cylindrical coordinate system for the Pinned constraint.

In previous releases, the software resolved the above condition using incorrect degrees of freedom. In this release, the software resolves this condition using a new constraint resolution rule: **Ignore Conflict**.

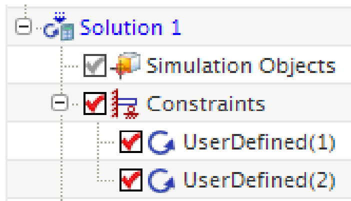
With this rule, the software applies the degrees of freedom values for both constraints to the common nodes.

You can also apply the new **Ignore Conflict** rule to other resolved constraint objects if you want the degrees of freedom of all the intersecting constraints to be taken into account for the common geometry or nodes.

The **Ignore Conflict** rule is used only when the intersecting constraints are both defined on the same type of object (for example, two edges, two faces, or two constraints on the same point). For other types of intersecting constraints (such as a face constraint and an edge constraint that share an edge or node), the appropriate resolution rule, such as **Use Object Precedence**, will be applied.

New workflow for resolving constraints

In Advanced Simulation, when the software detects a constraint conflict, it no longer initiates the constraint resolution automatically. Instead, icons now appear in the **Simulation Navigator** next to the conflicting constraints to indicate the conflict, as shown in the example below.



This feature alerts you to conflicting constraints and gives you control over resolving them appropriately. To resolve the constraints, do one of the following:

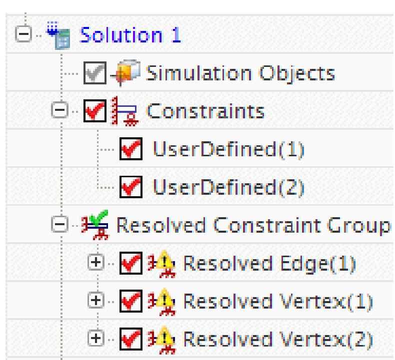
- In the **Advanced Simulation** toolbar, click **Update Finite Element Model**  .

  Note

  This button also updates any pending mesh updates.

- Right-click the solution node and choose **Resolve Constraints**.
- Use the **Exclude** option in the constraint dialog box to exclude the overlapping geometry and avoid the intersecting constraint.

As in previous releases, a **Resolved Constraint Group** node that contains the resolved constraint objects appears in the **Simulation Navigator**.



Enforced Displacement Constraints now available in axisymmetric structural solutions

What is it?

If Abaqus, ANSYS, or Nastran is your specified solver, you can now use the **Enforced Displacement Constraint** command in **Axisymmetric Structural** type solutions. You can use **Enforced Displacement Constraint** to apply a known displacement to your model.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with **Axisymmetric Structural** as the selected analysis type and Abaqus, ANSYS, NX Nastran, or MSC Nastran as the specified solver.
Toolbar	**Advanced Simulation→Enforced Displacement Constraint** 

Nastran

Bolt pre-loads now supported for Response Simulation analyses

What is it?

You can use the **Bolt Pre-Load** command to to apply a pre-load to a bolt modeled with CBAR or CBEAM type beam elements in a Response Simulation (**SEMODES 103 - Response Simulation**) analysis. Bolt pre-loads are only supported in the first two subcases in a Response Simulation:

- Static Offset
- Stress Stiffening

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran as the specified solver and **SEMODES 103 - Response Simulation** as the solution type and with either **Subcase - Static Offset** or **Subcase - Stress Stiffening** active in the **Simulation Navigator**
Toolbar	**Advanced Simulation→Bolt Pre-Load** 

Abaqus

Bolt pre-loads now supported for solid elements

What is it?

In the Abaqus solver environment, you can now define a pre-load on a bolt modeled with solid (continuum) elements. In previous releases, you could only define a bolt pre-load on a bolt modeled with beam (B31) elements.

In the **Bolt Pre-Load** dialog box, use the new **Force on 3D Elements** option in the **Type** list to define a pre-load on solid elements. With this option, you select either elements or faces that define a pre-tension section.

In Abaqus, a pre-tension section is defined as a surface inside the bolt that bisects the bolt. The software transmits the specified pre-load force across the pre-tension section by means of a pre-tension node that you specify. This pre-tension node must not be attached to any element in your model.

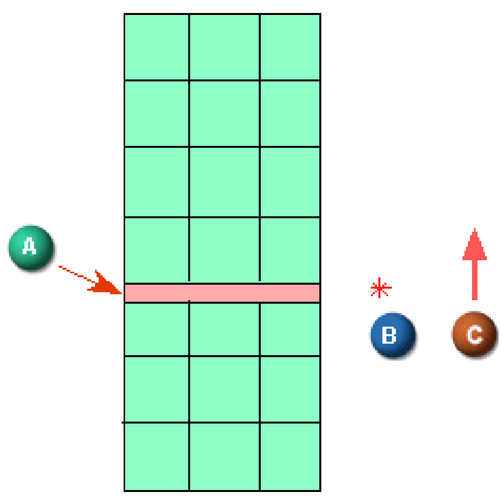
Note

If you do not specify a pre-tension node, the software creates a pre-tension node for you when you export or solve your model.

The software applies the load along a vector that is normal to the pre-tension section. The new **Section Normal** option lets you control how the software computes this normal.

- If you select **Average Surface Normal**, the software computes an average normal to the section that faces away from the underlying continuum elements.
- If you select **User Defined**, you can define the vector to specify the normal. This option is useful when the direction in which you want to apply the load is different from the average normal to the pre-tension section.

The following graphic shows an example of a bolt created with solid elements. (A) shows the pre-tension section, (B) shows the pre-tension node, and (C) shows the normal to the pre-tension section.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver
Toolbar	**Advanced Simulation→Bolt Pre-Load**

Initial clearance values for contact and bolt contact analyses

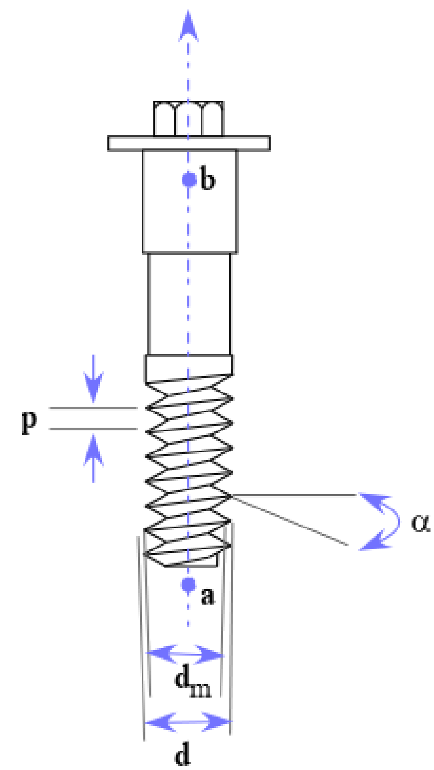
What is it?

You can use the new **Contact with Clearance** and **Bolt Contact with Clearance** simulation objects to define precise initial clearance or overclosure (initial penetration) values for the nodes on the slave (dependent) surface in a contact pair. With both **Contact with Clearance** and **Bolt Contact with Clearance**, the initial clearance or overclosure value you specify overwrites the initial clearance or overclosure value that the software calculates at each slave node.

- Use **Contact with Clearance** to define initial clearances or overclosures when you model contact between two surfaces.
- Use **Bolt Contact with Clearance** to define initial clearances or overclosures when you model contact between a single-threaded bolt and a bolt hole.

Contact with clearance for threaded bolts

The **Bolt Contact with Clearance** dialog box includes additional **Clearance Definition** options that let you model the thread characteristics of a bolt even if detailed thread geometry is not included in the model. These options let you specify details about the bolt threads, including the half thread angle (α), pitch (p, or the thread-to-thread distance), and major (d) and mean (m_d) bolt diameters. You also use the **Bolt Axis** options to define two points (a and b) along the bolt's axis. The software uses these points to generate the bolt's contact normal directions.



Clearance or overclosure value can be uniform or spatially varying

In the **Contact with Clearance** and **Bolt Contact with Clearance** dialog boxes, you can use the **Clearance Definition** options to define the clearance or overclosure value as either uniform or spatially varying for the contact pair. From the **Value** list:

- If you select **Expression**, you can specify a uniform clearance or overclosure value for the contact pair. A positive value indicates a clearance value, and a negative value indicates an overclosure value.
- If you select **Field**, you can specify spatially varying clearances or overclosures. With this option, you use a table field to specify the clearance at a single node or set of nodes on the slave surface. In the table field, the node ID is the independent variable, while the clearance or overclosure value is the dependent variable. You can also specify a **Scale Factor** to apply to the field.

Contact with clearance is supported only in small-sliding contact analyses

You can only use the **Contact with Clearance** or **Bolt Contact with Clearance** commands when you are using the small-sliding contact formulation in your analysis. In Abaqus, you use the *CONTACT PAIR keyword to specify the contact formulation. In Advanced Simulation, you use a **Contact Pair** modeling object to specify the parameters for the *CONTACT PAIR keyword:

1. In the **Contact Pair** dialog box, select **Small** from the **Sliding Type** list to use the small-sliding formulation instead of the finite-sliding formulation.
2. In the **Contact with Clearance** or **Bolt Contact with Clearance** dialog box, use the **Contact Pair** option to associate the **Contact Pair** modeling object with the simulation object.

Associated Abaqus keywords

When you export or solve your model, the software uses the options you specify in the **Contact with Clearance** dialog box to define the *CONTACT PAIR and *CLEARANCE keywords in your Abaqus input file. For more information, see *Adjusting Initial Surface Positions and Specifying Initial Clearances in Abaqus/Standard Contact Pairs* in the *Abaqus Analysis User's Manual* and **CLEARANCE* and **CONTACT PAIR* in the *Abaqus Keywords Reference Manual*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver
Toolbar	**Advanced Simulation→Contact with Clearance** or **Bolt Contact with Clearance**

New thermal conductance boundary condition

What is it?

When you are working with Abaqus as your solver, you can use the new **Surface-to-Surface Thermal Conductance** simulation object to model conductive heat transfer between proximate or contacting surfaces.

In the **Surface-to-Surface Thermal Conductance** dialog box, you can use the **Conductance Dependency** options to model the conductive heat transfer as a function of:

- The clearance between the contacting surfaces (**Clearance** option).
- The contact pressure at the interface between the contacting surfaces (**Pressure** option).
- Both the clearance and the contact pressure (**Clearance and Pressure** option).

In Advanced Simulation, you use fields to define how the heat transfer varies with the clearance and/or contact pressure.

When you export or solve your model, the software uses the options you specify in the **Surface-to-Surface Thermal Conductance** dialog box to define the *GAP CONDUCTANCE keyword in your Abaqus input file. For more information, see *Thermal Contact Properties* in the *Abaqus Analysis User's Manual* and **GAP CONDUCTANCE* in the *Abaqus Keywords Reference Manual*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver and **Thermal** as the selected analysis type
Toolbar	**Advanced Simulation→Surface-to-Surface Thermal Conductance**

ANSYS

Solver parameter to group face pressure loads

What is it?

A new **Create ANSYS components (CM) for similar loads** option has been added to the **Solver Parameters** dialog box. When you export or solve your model, this option controls whether the software uses the ANSYS CM command to group all elements that have the same face **Pressure** load into a component.

- If you select this option, the software uses the CM command to group all the elements with the same face **Pressure** load and only writes out a single face pressure (SFE) command in your ANSYS input file.
- If you clear this option, the software writes out individual single face pressure (SFE) commands for each element in your ANSYS input file.

You may want to select the **Create ANSYS components (CM) for similar loads** option if you plan to work with your ANSYS input file outside of Advanced Simulation. For example, if you select this option, you can later select the surface pressures by their component name in the ANSYS pre- and post-processing software. Additionally, the **Create ANSYS components (CM) for similar loads** option creates fewer SFE commands, so you may want to select the option to make manually editing your input file easier.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with ANSYS as the specified solver
Simulation Navigator	Right-click a solution→**Solver Parameters**

New options to control the deletion of loads at the end of a step

What is it?

A new **Delete load options** tab has been added to the **Create Solution Step** and **Edit Solution Step** dialog boxes. The options on this tab control whether forces and moments, body loads, and element body loads are kept or deleted by ANSYS at the end of the current solution step. These options correspond to the ANSYS DDELE, FDELE, BFDELE, and BFEDELE commands.

> **Note**
>
> Currently, you can only use these options to delete the loads or constraints on all nodes or elements to which they are applied.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Simulation Navigator	Either right-click a solution and select **Create Step**, or right-click the appropriate solution step and select **Edit Solution Step**

Cumulative loading options now preserved on import

What is it?

When you import an ANSYS input file into Advanced Simulation, the software now preserves the settings of any cumulative option commands (DCUM, FCUM, SFCUM, and BFCUM). In ANSYS, you use these commands to specify whether the software should add (ADD), replace (REPL), or ignore (IGNO) a repeated load or constraint on a particular degree-of-freedom. In previous releases, the software always imported any cumulative option command with a setting of “replace” (REPL), regardless of what you actually specified in the input file. Now, the software correctly preserves the specified cumulative option setting when you import the file.

> **Note**
>
> Advanced Simulation does not import any cumulative options as a **Solution Step** attribute. Instead, the software converts the load or boundary condition data into an equivalent load or boundary condition with the appropriate cumulative option setting.

For example, suppose you have the following constraints on node ID2:

- Step 1: UX = 1mm, UZ= 1.2mm
- Step 2: UX = 1.5mm, UY=1

In step 2 of the analysis, how the software resolves the constraint on node ID2 depends on the setting of the DCUM command:

- If the DCUM option is set to REPL, the resolved constraint for node ID2 is UX = 1.5, UY = 1 (UZ is removed)
- If the DCUM option is set to ADD, the resolved constraint for node ID2 is UX = 2.5, UY = 1, UZ = 1.2

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Menu	**File→Import→Simulation**

Support for importing loads and constraints from binary files

What is it?

When you import an ANSYS binary file into Advanced Simulation, the software now imports any loads (forces/moments) and constraints (displacements and temperatures) defined in that file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Menu	**File→Import→Simulation**

Groups

Node and element sets replaced by groups

What is it?

In this release, node and element sets have been completely replaced by groups. The **Sets** command has been removed from the product. Previously, you could use the **Node Set** and **Element Set** capabilities to create named collections of either nodes or elements. However, you could only create a set of a homogeneous collection of entities, and sets were only supported in a subset of the Advanced Simulation commands. Compared to sets, groups:

- Allow you to create named collections of heterogeneous types of entities.
- Are supported by a wider array of commands.
- Offer you greater flexibility and control over how and where they are created.

Commands now let you select groups instead of sets

All commands that previously allowed you to select node or element sets now allow you to select groups. For example, the Nastran **Structural Output Request** and **Thermal Output Request** modeling object dialog boxes have been modified to allow you to select groups of nodes or elements on which to output results.

Migration of existing node and element sets to groups

If you have a model from a previous release that contains node or element sets, the software automatically migrates those sets to groups when you open your model for the first time in this release.

Nastran SETS now imported as groups

In previous releases, if you imported a Nastran input file, any SET case control commands were imported as either a node set or as an element set. Now, the software imports SETs as groups. For details, see Understanding the import of Nastran SETs.

Grouping enhancements

What is it?

This release includes a number of enhancements to the grouping capabilities in Advanced Simulation.

Access to groups from the Simulation Navigator

A new **Groups** folder has been added the **Simulation Navigator**. The folder allows you to easily view and manage all existing groups. In the **Groups** folder, you can use right-click menu options to:

- Create new groups.
- Access the **Group Manager** dialog box where you can modify existing groups.

New Add to Group and Remove from Group commands

New **Add to Group**  and **Remove from Group**  commands are now available. You can access these commands from either the **Group Manager** dialog box or the right-click menu when you highlight a group in the **Simulation Navigator**. You can use these commands to quickly:

- Add new entities to a selected group.
- Remove selected entities from a selected group.

Initial support for group labels

When you create or import a group, Advanced Simulation now assigns a unique label (ID) to that group. Currently, you cannot change the label or specify criteria used to initially assign the label. The labels appear in the new **Label** column in the group list in the **Group Manager** dialog box. Because you can have groups defined in both the FEM and Simulation files, or in different FEM files within an assembly FEM, the software automatically resolves any conflict between the labels to enforce uniqueness.

Application	Advanced Simulation
Prerequisite	An active FEM, Simulation, or assembly FEM file
Simulation Navigator	Right-click a selected group.
Menu	**Format→Group→Manage**

Fields

General field enhancements

What is it?

This release includes several enhancements to fields. In Advanced Simulation, you use fields to define variant functions that specify magnitudes or properties in terms of other variables, such as time.

Fields now available from the Simulation Navigator

You can now manage fields from the **Simulation Navigator** as well as the **Part Navigator**. You can use the new **Fields** node in the **Simulation Navigator** to create, plot, edit, export, copy, rename, or delete a field. The **Fields** node is always visible, regardless of which Simulation part or file is currently active. Because fields have so many applications in Advanced Simulation, having the **Fields** node in the **Simulation Navigator** makes it easier for you to work with fields in the context of a CAE analysis.

Descriptions for fields

Description boxes are now available in the **Formula Field**, **Table Field**, and **Linked Field** dialog boxes. When you create a field, use the **Description** box to enter a description of the field.

Default bounds for independent domain variables

You can now define default bounding ranges for your named independent variables. These bounds are then used for all independent variables, so you do not need to define them each time you create a field, unless you want to change the default bounds. In the **Named Variables** dialog box, right-click a variable and choose **Default Bounds**. In the **Default Named Variable Bounds** dialog box, you can define the default **Minimum**, **Maximum**, **Default** value, and the **Number of Points**.

Ability to display spatial fields

What is it?

You can now create displays of the spatial fields in your model. Spatial fields define how a boundary condition or the thickness of elements map to an area of your model. Creating a display of a spatial field lets you visually verify that the distribution of the data is correct across the model.

Controlling the display of fields from the Simulation Navigator

You can control whether a field is displayed directly from the **Simulation Navigator**.

Fields


Temperature

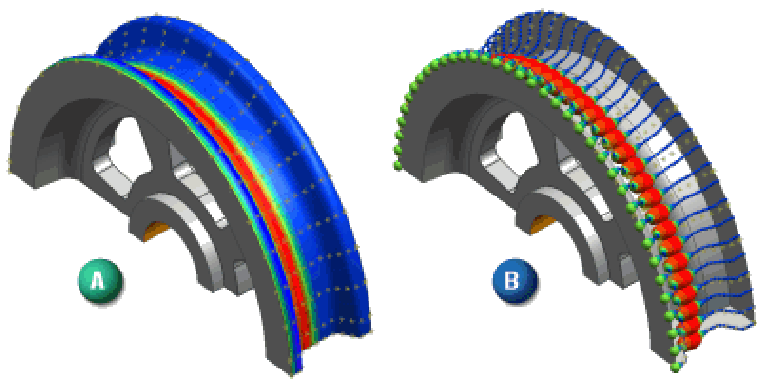
Force vs Time Formula

You can:

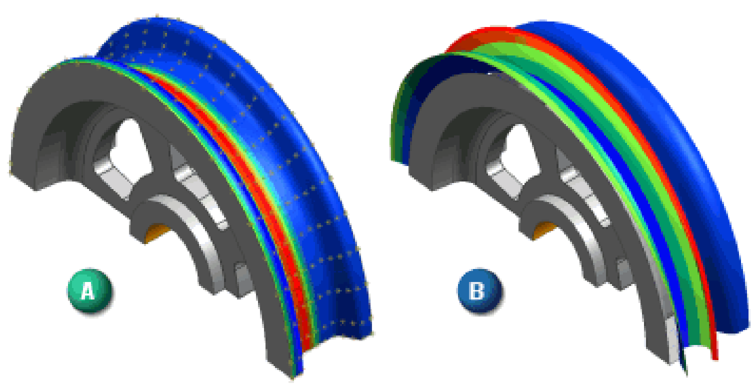
- Use the check boxes adjacent to individual fields to turn on or off the display of selected fields.
- Use the check box adjacent to the main **Fields** node to turn on or off the display of all fields.

Controlling the appearance of field displays

You can use the options in the new **Field Display Properties** dialog box to control how the software displays the spatial fields. For example, you can use the **Display Type** option in the **Dependent Domain** group to control the display of the data in the dependent field. Below, (A) shows a **Surface** type display of a spatial temperature field, while (B) shows a **Symbol** type display.



The available display options differ depending on whether you are working with the independent domain or the dependent domain. They also differ depending upon the type of display you create. For example, if you select **Surface** from the **Display Type** list in the **Dependent Domain** group, you can use the **Offset** option to control the degree to which the software offsets the data from the independent domain. Below (A) shows a surface with no offset, while (B) shows a surface with a much larger offset.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation or FEM file
Simulation Navigator	Right-click the field in the **Fields** node→**Edit Display**

Ability to map a field to an isoparametric plane or line

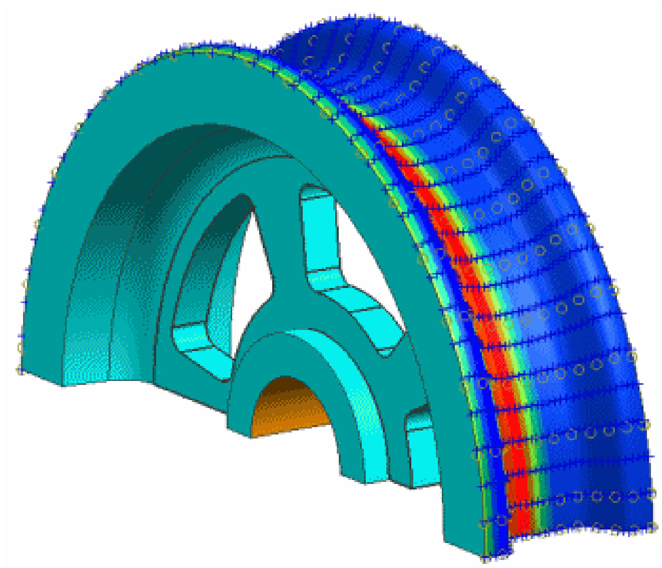
What is it?

Two new methods of mapping variable data to a location have been added. You can now map field data to an abstract combination of faces or lines in your model. You can then use this field as a boundary condition or as a variable thickness on the elements in your mesh.

Parametric Plane

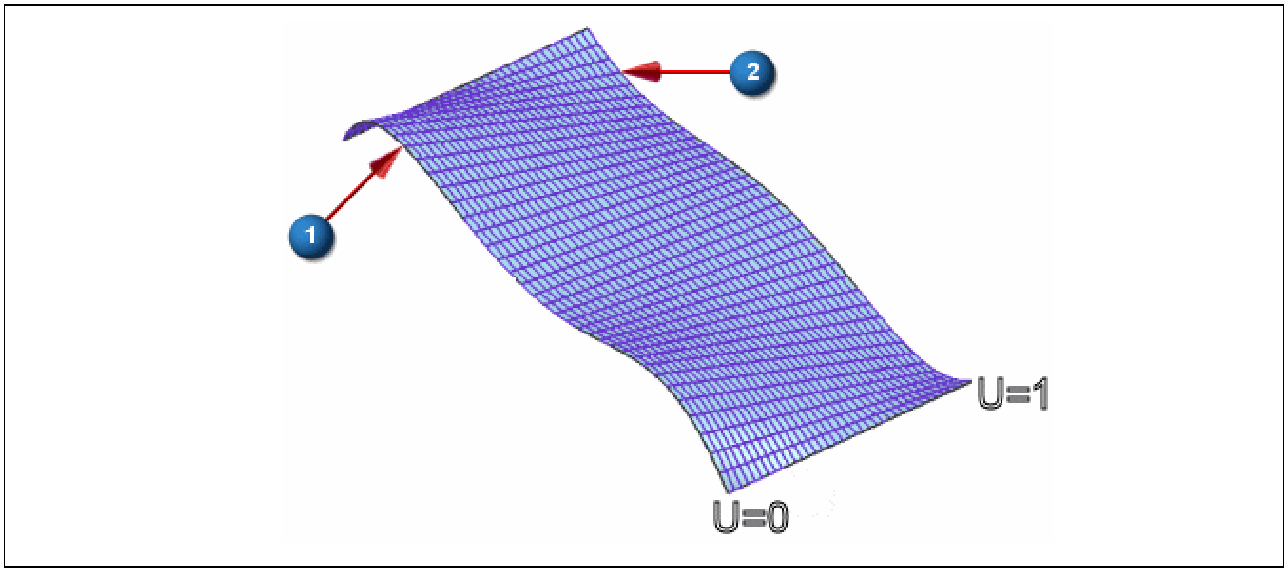
Using the new **Parametric Plane** spatial map type, available with the **Parameter Plane** spatial domain, you can map a field to a 2D arbitrary surface whose principal coordinate directions lie between 0,0 and 1,1. The coordinate directions are labeled U and V.

For example, you can use this method to define a varying temperature (for use in a boundary condition) or element thickness across a complex surface.

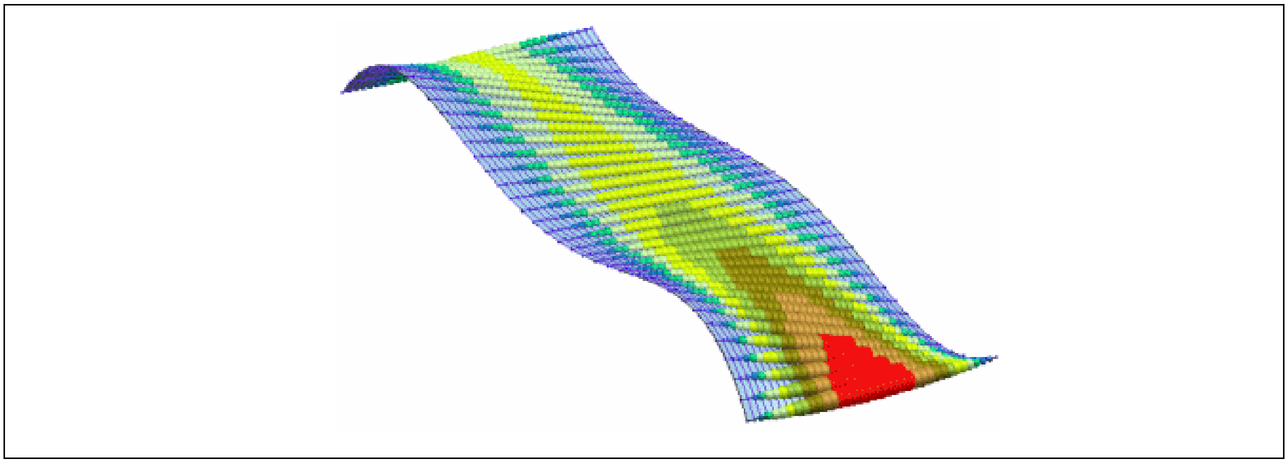


Temperature varying according to parametric plane

You define the U and V coordinates, which range from 0 to 1, by selecting points or edges in your model. You can define the U coordinates and allow the software to infer the V coordinates, or you can define both the U and V coordinates explicitly.



Example: U coordinates defined by selecting edge (1) and edge (2); V coordinates inferred



Data mapped to U and V coordinates

You can define the U and V coordinates using one of these methods:

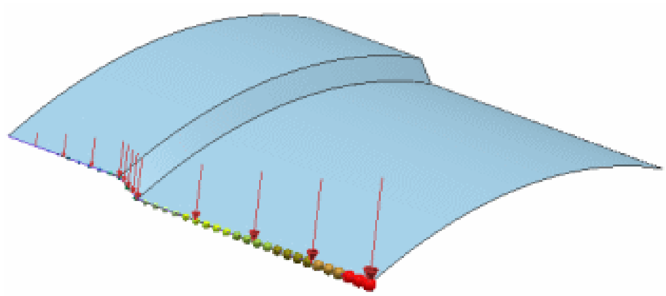
- **Iso Section** — You define at least two U “sections” by selecting points or edges in the model (the first section you define is 0% and the last one is 100%). The software creates the remaining U sections on the surface based on the sections you define. The software then infers the V sections from your U sections. This method works well when the geometry is linear in the direction of the V sections.
- **Iso Lines** — You define both the U and V sections. The definition process is the same as it is for the Iso Section mapping, except you define the V sections rather than letting the software infer them. This option is necessary when the model contains curved faces in the direction of the V sections

For more information, see Mapping a field to an isoparametric plane.

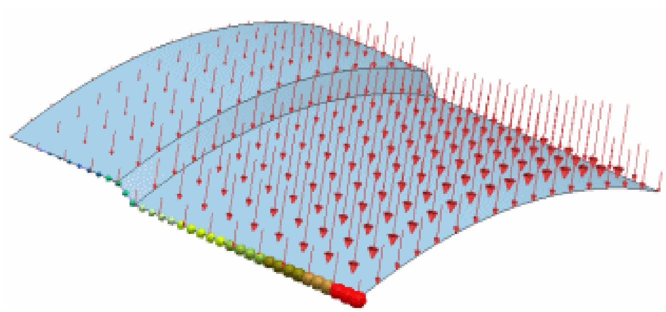
Parametric Line

Using the **Parametric Line** spatial map type, you can map variable data to a line. You can select points, edges, and curves to define a U section that defines a parametric line in Cartesian space. The software uses all objects you select to define a single curve with the starting point mapped to 0% and the end point mapped to 100%. This feature is useful for applying variable data spatially along an edge.

For example, you can use a parametric line field as a spatial scaling factor in a Force load:



This feature is also useful for defining a boundary condition or thickness that is varying in only one direction. For example, you can apply the above force distribution across the 2D faces using the same parametric line field:



For more information, see Mapping a field to an isoparametric line.

Where do I find it?

Parametric Plane

Application	Advanced Simulation
Prerequisite	An active Simulation or FEM file
Simulation Navigator	Right-click the **Fields** node→**New** [field type]
Location in dialog box	**Domain→Parameter Plane→Spatial Map→Type→Parametric Plane**

Parametric Line

Application	Advanced Simulation
Prerequisite	An active Simulation or FEM file
Simulation Navigator	Right-click the **Fields** node→**New** [field type]
Location in dialog box	**Domain→Parameter→Spatial Map→Type→Parametric Line**

Geometry idealization and abstraction

Midsurface improvements

What is it?

This release includes several improvements to the **Midsurface** command.

Improved usability for Midsurface dialog box

This release includes enhancements to the **Midsurface** dialog box to improve its usability.

- In previous releases, the **Midsurface** dialog box closed automatically after you generated a single midsurface. Now, the dialog box remains active until you explicitly close it.
- If you select the **Automatic Progression** check box, you can now create midsurfaces on multiple bodies without exiting the **Midsurface** dialog box. If you select **Automatic Progression**, when you generate a midsurface on a body, the software automatically displays only the new sheet body in the graphics window. In a model that contains multiple solid bodies, this helps you visually distinguish between the bodies that have defined midsurfaces and those that do not. You can then more easily select the next solid body on which to create a midsurface.

Improvements to automatic pairing and overall performance

When you use the **Face Pair** method with the **Auto Create** option, improvements have been made to the algorithm the software uses to automatically pair faces. On certain types of parts, the software now produces more accurate face pairs as well as more accurately trimmed surfaces.

Additionally, there are significant performance improvements in the time it takes the software to generate a midsurface on certain types of parts. These improvements occur when you create a midsurface with the **Advanced Creation and Trimming** check box cleared.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active idealized part
Toolbar	**Advanced Simulation→Midsurface** 

Split Body creates mesh mating conditions

What is it?

When you use the **Split Body** command to divide geometry into multiple bodies, use the new **Auto Create Mesh Mating Conditions** option in the **Split Body** dialog box to create mesh mating conditions between those bodies. With this option, when you switch from the idealized part to the FEM file, the software automatically creates a glue coincident mesh mating condition between the bodies. The mesh mating condition ensures that the meshes are continuous from one body to the other. Previously, you had to manually create mesh mating conditions between any bodies created by the **Split Body** command.

> **Note**
>
> If you use the **Split Body** command in the Modeling application, the **Auto Create Mesh Mating Conditions** option is called **Keep Imprinted Edges**. If you select **Keep Imprinted Edges**, the software creates mesh mating conditions between the bodies when you switch to the Advanced Simulation application.

Where do I find it?

Application	Advanced Simulation
Prerequisite	The idealized part active
Toolbar	**Advanced Simulation→Split Body** 

Split Edge improvements

What is it?

The **Split Edge** command now includes options that you can use to define the location where the software divides the edge. To facilitate these improvements, the **Split Edge** dialog bar available in previous releases has been converted to a new **Split Edge** dialog box.

The **Type** menu lets you select the method to use to split the edge:

- Use **Location on Edge** to divide the edge at a point you select. In previous releases, this was the only available method for splitting an edge.
- Use **Project Point on Edge** to divide the edge by projecting a point onto the edge. The software uses a closest point method to project the selected point to the edge. The software splits the edge at the location of the projected point.
- Use **Project Point Along a Vector** to divide the edge by projecting a point onto the edge along a specified vector. The software splits the edge at the location that represents the closest point projection of the point along the vector.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Model Cleanup→Split Edge** 

Materials

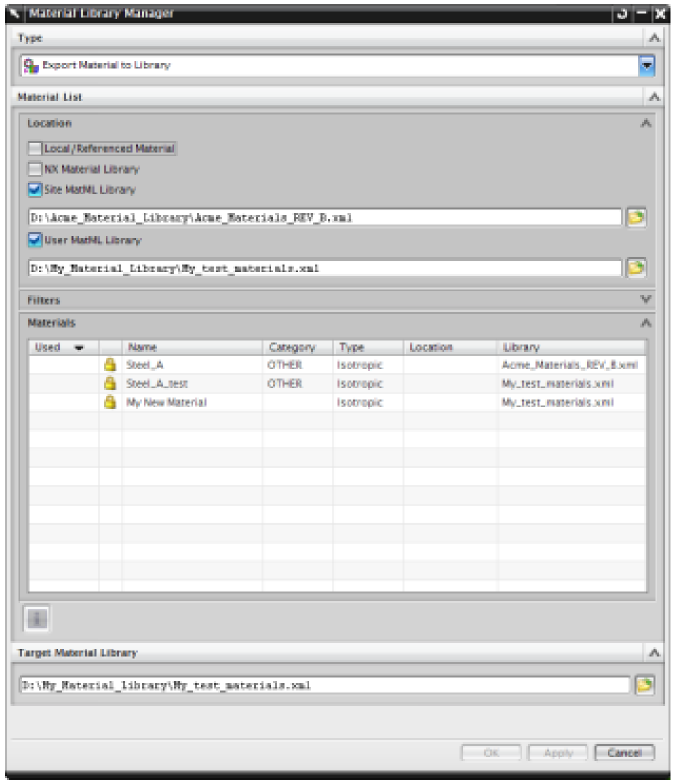
Support for external material libraries

What is it?

The process for using an external, custom material library in NX has been improved. You can now:

- Maintain two separate custom material libraries. Custom libraries are stored as XML files in the MatML schema version 3.1.5. For schema details, see http://www.matml.org.
- Restrict display and editing access to the two custom libraries independently.
- Create, edit, or delete your custom library materials directly within NX.
- Export local materials (materials created in NX and saved with a model) to a custom material library.
- Import material libraries generated from external sources that adhere to the MatML schema.

The new **Material Library Manager** dialog box lets you export materials to a custom library and edit or delete the custom materials.



Use customer defaults for the Materials function to define the default folder location of the material libraries, as well as configure access to each library.

Why should I use it?

- Custom material definitions can be used in multiple NX models, and these custom materials are preserved when you upgrade to a newer version of NX.
- Editing your materials in the NX user interface is much easier than editing the phys_material.dat file.

Where do I find it?

Materials customer defaults

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Gateway→General→Materials** tab

Material Library Manager command

Application	All
Menu	**Tools→Material Library Manager**
Simulation Navigator	**Advanced Simulation** toolbar→**Material Library Manager**

Enhancements for isotropic and fluid materials

What is it?

As part of the ongoing enhancements to the NX materials capabilities, isotropic and fluid materials now support a number of functional enhancements. With these materials, you can now:

- Enter the value for a material property as an Expression (that is, a formula, function, reference, or constant value) or as a Field (a table or formula).
- Plot tabular material property values using the XY Graphing capability.
- Specify units for any material property. For example, you can set the units for a stress-strain curve.
- Enter a description for any material.
- View all material properties in the **Information** view for a given material.

In addition, the user interface for creating new isotropic or fluid materials has been improved for better usability.

Orthotropic and anisotropic materials will offer the same capabilities in a future release.

Where do I find it?

Application	All
Menu	**Tools→Material Properties**
Simulation Navigator	**Advanced Simulation** toolbar→**Material Properties**

Material damping support

What is it?

Material damping is now supported for relevant Nastran and ANSYS materials. This allows you to include structural damping in dynamic analyses.

- For Nastran, the GE field is now supported for the MAT1, MAT2, MAT3, MAT8, and MAT9 bulk data entries.
- For ANSYS, the DAMP option is now supported for the MP command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM, idealized part, or part
Toolbar	**Advanced Simulation→Material Properties** 

Nastran

New hyperelastic material models

What is it?

This release includes support for new NX Nastran hyperelastic material models. Hyperelastic materials let you model materials that are nearly incompressible but which undergo large strains.

Hyperelastic Material Model	Corresponding ANSYS Command
Arruda-Boyce	MATHE bulk data entry with Model field = Aboyce
Foam	MATHE bulk data entry with Model field = Foam
Mooney-Rivlin	MATHE bulk data entry with Model field = Mooney
Ogden	MATHE bulk data entry with Model field = Ogden

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM, idealized part, or part with NX Nastran as the specified solver
Toolbar	**Advanced Simulation→Material Properties** 

Abaqus

New hyperelastic material models

What is it?

This release includes support for an expanded range of Abaqus hyperelastic material models, including several models in which you can use test data to define certain material properties. This release also includes support for a new **Gasket Behavior** material.

- Hyperelastic materials let you model materials that are nearly incompressible but which undergo large strains.
- The Abaqus **Gasket Behavior** material lets you model gasket behavior properties, such as elastic properties for the membrane and transverse shear behaviors of a gasket, for gasket analyses.

New materials

Material	Corresponding Keyword
Arruda-Boyce	*HYPERELASTIC keyword with both the ARRUDA-BOCYE parameter (constants option)
Arruda-Boyce Test Data	*HYPERELASTIC keyword with the ARRUDA-BOCYE and TEST DATA INPUT parameters
Foam	*HYPERFOAM keyword (constants option)
Foam Test Data	*HYPERFOAM keyword with the TEST DATA INPUT parameter
Gasket Behavior	*GASKET BEHAVIOR keyword
Marlow	*HYPERELASTIC keyword with the MARLOW parameter
Mooney-Rivlin	*HYPERELASTIC keyword with the MOONEY-RIVLIN parameter (constants option)
Neo Hooke	*HYPERELASTIC keyword with the NEO HOOKE parameter (constants option)
Neo Hooke Test Data	*HYPERELASTIC keyword with both the NEO HOOKE and TEST DATA INPUT parameters
Ogden	*HYPERELASTIC keyword with the OGDEN parameter (constants option)
Ogden Test Data	*HYPERELASTIC keyword with the OGDEN and TEST DATA INPUT parameters
Polynomial	*HYPERELASTIC keyword with the POLYNOMIAL parameter (constants option)
Reduced Polynomial	*HYPERELASTIC keyword with the REDUCED POLYNOMIAL parameter (constants option)
Van Der Waals	*HYPERELASTIC keyword with the VAN DER WAALS parameter
Yeoh	*HYPERELASTIC keyword with the YEOH parameter

	(constants option)
Yeoh Test Data	*HYPERELASTIC keyword with both the YEOH and TEST DATA INPUT parameters

Defining material constants using test data

With the test data material types, you can use a field to specify the deformation modes that define the material constants with test (experimental) data.

- Use the **UNIAXIAL Tension/Compression** option to specify uniaxial test data. This option corresponds to the Abaqus *UNIAXIAL TEST DATA keyword. With **UNIAXIAL Tension/Compression**, you define a table field in which you list the material's nominal stress (T_U) and nominal strain values (ε_U) on each line.
- Use the **BIAXIAL Tension** option to specify biaxial test data. This option corresponds to the Abaqus *BIAXIAL TEST DATA keyword. With **BIAXIAL Tension**, you define a table field in which you list the material's nominal stress (T_B) and nominal strain values (ε_B) on each line.
- Use the **PLANAR - Pure Shear** option to specify planar (or pure shear) data. This option corresponds to the Abaqus *PLANAR TEST DATA keyword. With **PLANAR - Pure Shear**, you define a table field in which you list the material's nominal stress (T_S) and nominal strain in the direction of loading (ε_S) on each line.
- Use the **Pure Volumetric Compression** option to specify volumetric loading test data to include user-defined material compressibility. With **Pure Volumetric Compression**, you define a table field in which you list the material's pressure (p) and the volume ratio, J (current volume/original volume) on each line.

Depending upon the material's type, you can use one or more of these options to define the experimental stress-strain data.

- With the **Arruda-Boyce Test Data**, **Marlow**, and **Foam Test Data** materials, you can only use one of the test data options to define the experimental test data. If you use more than one, when you export or solve your model, the software only writes out the first applicable test data curve. The order in which the software searches for the appropriate test data option depends on the material type, as follows:
  - Arruda-Boyce Test Data: **BIAXIAL Tension** then **UNIAXIAL Tension/Compression**
  - Marlow: **BIAXIAL Tension**, then **PLANAR - Pure Shear**, then **UNIAXIAL Tension/Compression**
  - Foam Test Data: **PLANAR - Pure Shear**, then **UNIAXIAL Tension/Compression**, then **Pure Volumetric Compression**
- With the **Mooney-Rivlin Test Data**, **Neo Hooke Test Data**, **Ogden Test Data**, **Reduced Polynomial**, **Van Der Waals**, and **Yeoh Test Data** material types, you can use up to four of the options to define the experimental test data. If you use more than one, when you export or solve your model, the software writes out the test data option in the following order: **BIAXIAL Tension**, then **PLANAR - Pure Shear**, then **UNIAXIAL Tension/Compression**, then **Pure Volumetric Compression**.

For more information, see:

- **HYPERELASTIC*, **UNIAXIAL TEST DATA*, **BIAXIAL TEST DATA*, **PLANAR TEST DATA*, and **VOLUMETRIC TEST DATA* in the *Abaqus Analysis Keywords Manual*.
- *Hyperelastic behavior of rubberlike materials* in the *Abaqus Analysis User's Manual*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM, idealized part, or part with Abaqus as the specified solver
Toolbar	**Advanced Simulation→Material Properties**

ANSYS

New hyperelastic material models

What is it?

This release includes support for a number of new ANSYS hyperelastic material models. Hyperelastic materials let you model materials that are nearly incompressible but which undergo large strains.

Hyperelastic Material Model	Corresponding ANSYS Command
Arruda-Boyce	TB command with the HYPER,,,,BOYCE option
Foam	TB command with the HYPER,,,,FOAM or HYPER,,,,BLATZ options
Gent	TB command with the HYPER,,,,GENT option
Mooney-Rivlin	TB command with the HYPER,,,,MOONEY option
Neo Hooke	TB command with the HYPER,,,,NEO option
Ogden	TB command with the HYPER,,,,OGDEN option
Polynomial	TB command with the HYPER,,,,POLY option
Yeoh	TB command with the HYPER,,,,YEOH option

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM, idealized part, or part with ANSYS as the specified solver
Toolbar	**Advanced Simulation→Material Properties**

Meshing

Newly supported element types

What is it?

This release includes support for additional NX Nastran and Abaqus element types.

New NX Nastran Element Type

Advanced Simulation now supports the CPYRAM element. For more information, see:

- **Unsatisfied xref title** for details on how to include pyramid elements in your model.
- *CPYRAM* in the *NX Nastran Quick Reference Guide*

New Abaqus Element Types

This release includes support for the following types of Abaqus gasket elements:

Abaqus Element	Description
GK3D6	6-noded, three-dimensional gasket element
GK3D6N	6-noded, three-dimensional gasket element with thickness-direction behavior only
GK3D8	8-noded, three-dimensional gasket element
GK3D8N	8-noded, three-dimensional gasket element with thickness-direction behavior only

How you create the gasket elements depends upon the element's type:

- To create GK3D8 or GK3D8N type elements, use the **3D Swept Mesh** command.
- To create GK3D6 or GK3D6N type elements, first create a 2D mesh of either linear or parabolic triangular elements. Then, use the **3D Swept Mesh** command to generate the GK3D6 or GK3D6N elements.

For more information on modeling and analyzing gaskets, see **Unsatisfied xref title**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with the appropriate solver specified
Toolbar	**Advanced Simulation** toolbar→ **3D Tetrahedral Mesh**  or **3D Swept Mesh** 
Menu	**Insert→Mesh→3D Tetrahedral Mesh** or **3D Swept Mesh**

Improved tetrahedral meshing algorithm

What is it?

This release includes improvements to the tetrahedral meshing algorithm. The most significant of these is how the algorithm manages memory during mesh generation. Specifically:

- The algorithm no longer has a contiguous memory requirement. Previously, if the memory on your machine was fragmented, the meshing algorithm could only utilize the largest contiguous free block of memory. Now, the software utilizes all available memory. This means that the software can now take advantage of all available smaller blocks of memory on your machine.
- The algorithm's memory allocation is now tuned for 64-bit precision. Because the algorithm has been tuned to take advantage of the increased precision, the overall robustness of the algorithm has increased. Previously, the software could allocate an array past the 32-bit limit, but all the floating point calculations were still performed with 32-bit floats.

These improvements are most noticeable when you generate meshes on large models or when you iteratively reduce the element size on a smaller model.

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**

Quadrilateral only meshing

What is it?

You can use new options in the **2D Mesh**, **2D Mapped Mesh**, and **3D Swept Mesh** dialog boxes to create a quadrilateral only mesh. A quadrilateral only mesh is a special class of free mesh that does not contain any triangular elements. Quadrilateral only meshes are sometimes necessary in analyses in which the presence of triangular elements is considered either undesirable or unacceptable.

Quadrilateral only 2D free meshes

Use the new **Attempt Quad Only** option in the **2D Mesh** dialog box to control whether the software tries to generate a mesh that contains only quadrilateral elements. With a quadrilateral only mesh:

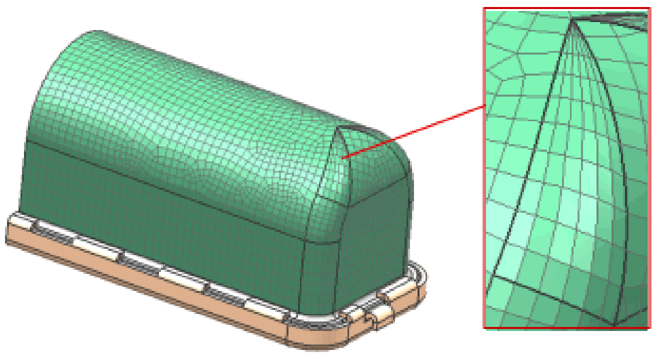
- If you select **On – Zero Triangles**, the software creates a mesh that does not contain any triangular elements. With this option, if the software cannot create a mesh with only quadrilateral elements, it does not generate a mesh.
- If you select **On – Minimum Triangles**, the software creates a mesh that contains, at most, a single triangular element per selected face. With this option, the software only inserts a triangle if it cannot establish nodal parity (an even number of nodes) along the boundary of the face.

To create a quadrilateral only mesh, you must first clear the **Attempt Free Mapped Mesh** check box.

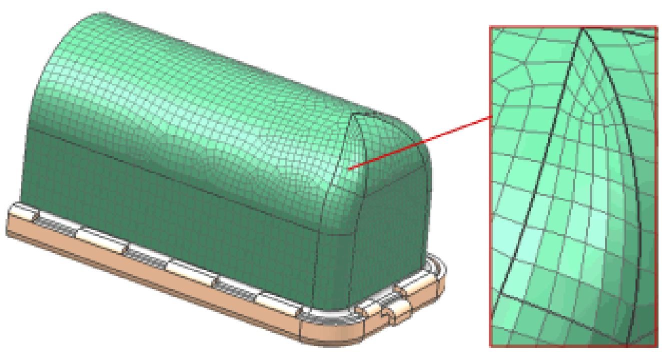
Quadrilateral only 2D mapped meshes for three-sided faces

Use the new **Quad Only on 3 Sided Faces** option in the **2D Mapped Mesh** dialog box to create a mesh that does not contain any triangular elements on a three-sided face. If you select this option, the software tries to generate an all quadrilateral mesh for faces that have only three sides. With this option, if the software is unable to maintain nodal parity due to constraints from existing meshes on the surrounding faces, it creates a mapped mesh with a single triangular element. In previous releases, if you created a mapped mesh on a three-sided face, the mesh always degenerated into triangular elements at a selected corner.

The following graphic shows a mapped mesh generated with the **Quad Only on 3 Sided Faces** check box cleared. Notice how the quadrilateral mesh degenerates into triangular elements at the uppermost corner.



This graphic shows a mapped mesh on the same geometry generated with the **Quad Only on 3 Sided Faces** check box selected. This mesh is comprised of only quadrilateral elements.



Quadrilateral only source faces for 3D hexahedral meshes

Use the new **Attempt Quad Only** options in the **3D Swept Mesh** dialog box to have the software generate a quadrilateral only mesh on the source face. Having an all quadrilateral element mesh on the source face ensures that the swept mesh through the volume will not contain any wedge elements.

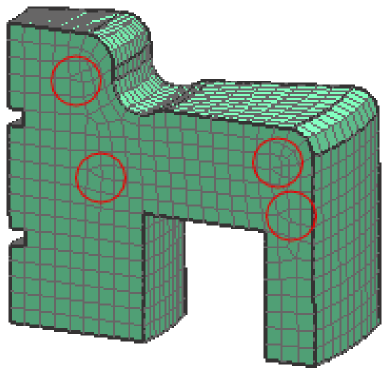
Use the options in the **Attempt Quad Only** list to control whether the software creates a mesh on the source face that either does not include any triangles or that only contains one triangle.

With the **2D Mesh** dialog box, the **Attempt Free Mapped Mesh** and **Attempt Quad Only** options are mutually exclusive. However, with the **3D Swept Mesh** dialog box, if you select both **Attempt Free Mapped Mesh** and either **On – Zero Triangles** or **On – Minimum Triangles** from the

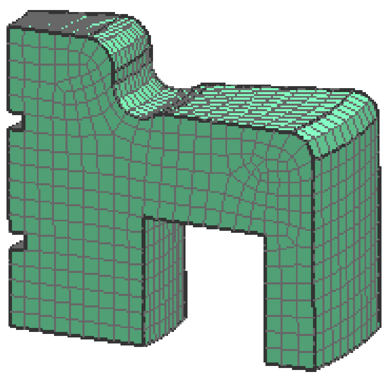
Attempt Quad Only list, the software proceeds through the options sequentially. First, it tries to generate a free mapped mesh. If that mesh fails, it then tries to create a quadrilateral only mesh. The software retains the first successful mesh it generates.

If the software cannot successfully generate either a quadrilateral only mesh or a free mapped mesh, it uses the **Paver** meshing method to generate a free mesh on the source face.

The following graphic shows a hexahedral mesh generated with the **Attempt Free Mapped Mesh** check box selected and the **Attempt Quad Only** option set to **Off**. Notice that the mesh on the source face contains several triangular elements.



This graphic shows a hexahedral mesh on the same part generated with the **Attempt Free Mapped Mesh** check box cleared and the **Attempt Quad Only** option set to **On – Zero Triangles**.



New customer defaults

New customer defaults let you control the default settings for the **Attempt Quad Only** options in the **2D Mesh** and **3D Swept Mesh** dialog boxes and the **Quad Only on 3 Sided Faces** option in the **2D Mapped Mesh** dialog box. To set these defaults:

1. From the **File** menu, choose **Utilities→Customer Defaults**.
2. In the **Customer Defaults** dialog box, choose **Simulation→Meshing**.
3. Click the **2D Free Mesh**, **2D Mapped Mesh**, or **3D Mapped Mesh page** and set the appropriate option.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation→2D Mesh**, , **2D Mapped Mesh** , or **3D Swept Mesh**
Menu	**Insert→Mesh→2D Mesh**, **2D Mapped Mesh**, or **3D Swept Mesh**

Creating a surface coat of 2D elements

What is it?

Use the new **Surface Coat** command to create a surface coating of 2D elements on top of existing 3D (solid) elements. You can create a surface coat on either the free faces of selected solid elements or on an entire solid mesh. The resulting 2D mesh:

- Uses the nodes and the connectivity of the solid mesh on which it is defined.
- Can be linear or parabolic if the solid mesh is parabolic. For example, you can create a surface coating of either linear or parabolic quadrilateral elements on a mesh of parabolic quadrilateral elements. However, if the solid mesh is linear, then the resulting 2D mesh can only be linear.
- Is not associated to the underlying 3D mesh. If you delete or modify either the 2D mesh or the 3D mesh, the other mesh is unaffected.
- Is not associated to the underlying geometry.

There are a number of different reasons why you might want to create a surface coating of 2D elements on your model. For example, you can create a surface coating of 2D elements to:

- Model an actual surface coating on the underlying part.
- Add mass to your model.
- Facilitate post-processing in cases, for example, where you are chiefly interested in the stress results on the surface of the part.
- Ensure that loads are correctly transferred between adjacent shell and solid meshes. For example, imagine a model that contains a thin rib that intersects the main body of a solid part. The rib is meshed with 2D elements, with 6 degrees-of-freedom each, while the part itself is meshed with 3D elements, with 3 degrees-of-freedom each. Suppose you need to apply a bending load to the end of the rib. To ensure that the moments transfer correctly from the 2D elements to the 3D elements, you can create a surface coating of 2D elements on the 3D elements that surround the location where the rib intersects the part.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains a 3D mesh
Toolbar	**Element Operations→Surface Coat** 
Menu	**Insert→Element→Surface Coat**

Modifying the order of elements in a mesh

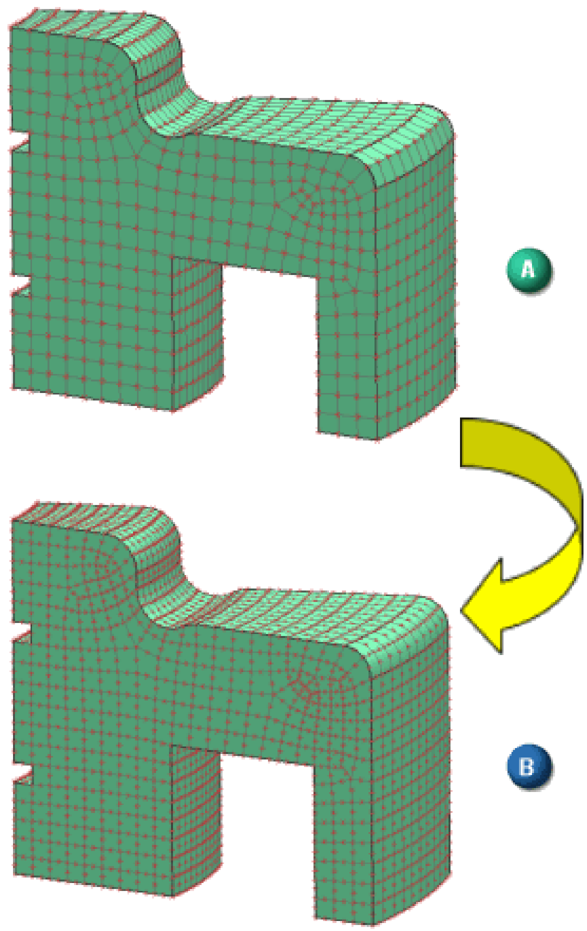
What is it?

Use the new **Element Modify Order** command to change the order of all elements in a selected mesh. For meshes that are associated to their underlying geometry, **Element Modify Order** lets you modify the element order without regenerating the entire mesh. You can change:

- A mesh of parabolic elements to a mesh of linear elements.
- A mesh of linear elements to a mesh of parabolic elements.

Modifying the order of elements with the **Element Modify Order** command is faster than changing the order of elements by editing an existing mesh.

In the following graphic, we used **Element Modify Order** to modify a mesh of linear hexahedral elements (A) to a mesh of parabolic hexahedral elements (B).



When you use **Element Modify Order**:

- The software does not change either the location or label of the corner nodes in the mesh.
- In a mesh that is associated to the underlying geometry, any midnode that the software inserts along an edge or face is associated to that edge or face.

Note

You cannot use **Element Modify Order** to modify the order of either a pyramid element mesh or a hexahedral or tetrahedral mesh that adjoins a pyramid element mesh.

Modifying the Midnode Method for parabolic elements

For a mesh of parabolic elements, you can also use **Element Modify Order** to change the specified **Midnode Method** for the mesh. The **Midnode Method** lets you control how the software projects the elements' midnodes onto the geometry.

For example, suppose you select **Mixed** from the **Midnode Method** list when you create the mesh. With the **Mixed** option, the software projects the midnodes on an element to the geometry unless the projection would cause the element's Jacobian value to exceed the **Maximum Jacobian** threshold specified in the **2D Mesh** dialog box. You could later use **Element Modify Order** to change the method to either **Curved** or **Linear**. However, if you select a parabolic mesh that is not associated to the underlying geometry, such as a mesh created with commands on the **Element Operations** toolbar, you can only change the **Midnode Method** to **Linear**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains a 2D or 3D mesh
Toolbar	**Element Operations→Element Modify Order** 
Menu	**Edit→Element→Modify Order**

Varying the thickness of 2D elements

What is it?

This release includes several enhancements that allow you to vary the thickness of 2D elements. You can:

- Use options in the **Mesh Associated Data** dialog box to vary the thickness of all the 2D elements in a mesh.
- Use options in the **Element Associated Data** dialog box to vary the thickness values within an individual element.

Improved user interface for defining the source of a 2D element's thickness

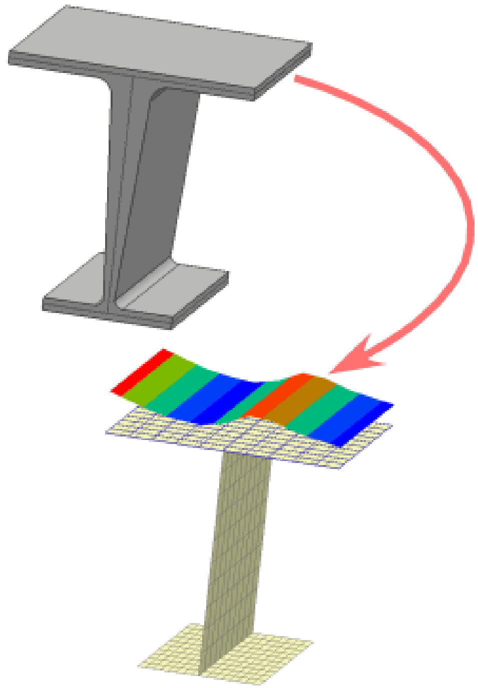
In the **Mesh Associated Data** dialog box, the **Ignore Midsurface Thickness** option from previous releases has been replaced by the new **Thickness Source** list. Use the **Thickness Source** list to specify the source of the data the software uses to assign the elements' thickness values. For example, if you select **Physical Property Table**, the software applies the thickness value defined in the associated physical property table to the mesh. In contrast, if you select **Midsurface**, the software applies the thickness of the midsurface to the mesh.

Ability to use fields to vary the thickness of elements across a mesh

In the **Mesh Associated Data** dialog box, the **Field** option in the **Thickness Source** list lets you use a spatial field to vary the thickness of the elements across a mesh. You can use:

- A formula field to vary the thickness of the elements using an NX expression (mathematical function).
- A table field to vary the thickness of the elements using a table of independent and dependent variables. With a table field you can import an external comma separated (.csv) text file that defines the appropriate thickness values to use as input for the table.

The following graphic shows a part with a 2D mesh generated on the part's midsurface. A table field defines the thickness of the elements along the top surface.



The following table lists the types of elements for each solver for which you can create a spatial field to vary the thickness.

Solver	Element Types that Support Thickness Fields
Nastran	CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR
Abaqus	SHELL63, SHELL93, SHELL181, PLANE42, PLANE82
ANSYS	S3, S3R, STRI3, STRI65, S4, S4R5, S8R, S8R5

Ability to specify different nodal thickness values for a single element

Use the new **Corner Node Thickness** options in the **Element Associated Data** dialog box to specify different thickness values for selected corner nodes. Any thickness values you specify with the **Element Associated Data** dialog box take precedence over the thickness value assigned to the overall mesh.

Where do I find it?

Mesh Associated Data

Application	Advanced Simulation
Prerequisite	An active FEM file
Menu	**Tools→Mesh Associated Data**
Simulation Navigator	Right-click the appropriate mesh→**Edit Mesh Associated Data**

Element Associated Data

Application	Advanced Simulation
Prerequisite	An active FEM file
Menu	**Edit→Element→Modify Associated Data**
Toolbar	**Element Operations→Element Associated Data**

New method for orienting 2D element normals

What is it?

Use the new **2D Element Normal by Seed** option in the **Model Check** dialog box to orient the normals of 2D elements to match the normal of a selected seed element. You can:

- Change the normals of all connected elements in the mesh to which the seed element belongs.
- Change the normals of all connected elements in all currently displayed 2D elements.

With **2D Element Normal by Seed**, you select an element whose orientation is in the desired direction. When you click either **OK** or **Apply**, the software modifies the elements' orientation.

You can use **2D Element Normal by Seed** to quickly modify the normals of elements so that they are consistent. Consistent element normals are important, for example, to ensure the proper post-processing of element stress and strain results.

> **Note**
>
> If you want your selection to stop at the end of a non-manifold edge, you should first merge any coincident nodes before you use the **2D Element Normal by Seed** option. To merge coincident nodes, select the **Nodes** option in the **Model Check** dialog box and then select **Merge Duplicate Nodes**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains a 2D mesh
Toolbar	**Advanced Simulation→Finite Element Model Check**

Initial pyramid element support for NX Nastran

What is it?

This release contains the initial implementation of pyramid element support for NX Nastran models. In Advanced Simulation, if you are working in the NX Nastran solver environment, you can now use pyramid elements to transition between hexahedral and tetrahedral meshes on adjacent bodies (volumes).

You can also import existing NX Nastran models that contain pyramid elements and work with those elements in Advanced Simulation. Previously, pyramid elements were only supported in the ANSYS solver environment.

NX Nastran uses the CPYRAM bulk data entry to define the pyramid elements. The CPYRAM element is only supported in Advanced Nonlinear (SOL 601 and SOL 701) analyses. A CPYRAM element can have 5 or 13 nodes. For more information, see *CPYRAM* in the *NX Nastran Quick Reference Guide*.

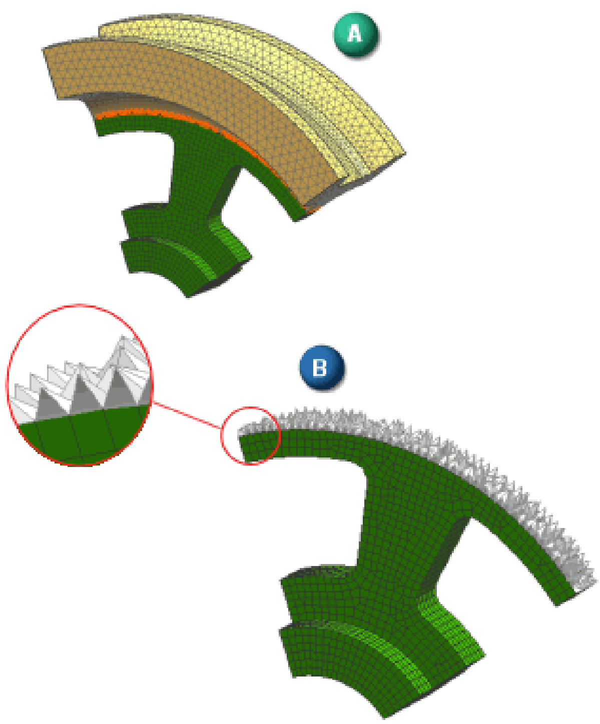
Creating pyramid elements at hexahedral-tetrahedral interfaces

When you generate a tetrahedral mesh on a body (volume) that is adjacent to a body with an existing hexahedral mesh, you can use the **Use Pyramids for Transition** option in the **3D Mesh** dialog box to have the software automatically create transitional pyramid elements between the meshes. For the software to create transitional pyramid elements:

- The adjacent body must have an existing mesh of hexahedral elements.
- A **Glue Coincident** type of **Mesh Mating Condition** must be defined at the interface between the two bodies to ensure that the nodes on the adjacent hexahedral and tetrahedral meshes match exactly.

For more information about using pyramid elements with the NX Nastran environment, see Using pyramid transitions with NX Nastran in the Advanced Simulation help.

The following example shows transitional pyramid elements at the interface of hexahedral and tetrahedral meshes on a section of a wheel hub. The outer section of the hub is meshed with tetrahedral elements, while the inner section is meshed with hexahedral elements. (A) shows the two meshes together, with the transitional pyramid elements highlighted in orange. (B) shows a view of just the pyramid elements with the tetrahedral elements.



Why should I use it?

Pyramid elements ensure a conforming mesh as they provide a direct transition from hexahedral elements to tetrahedral elements. In contrast, interface connection methods rely on either rigid elements or multi-point constraint equations to connect the nodes. Using pyramid elements to join dissimilar meshes offers advantages in greater solution accuracy and reduced solution time compared to rigid elements or multi-point constraint equations.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with NX Nastran as the specified solver
Toolbar	**Advanced Simulation→3D Tetrahedral Mesh** 
Menu	**Insert→Mesh→3D Tetrahedral Mesh**

New option for retaining free meshes in mapped mesh failures

What is it?

In previous releases, if the software was unable to generate a mapped mesh on one or more of the faces you selected with the **2D Mapped Mesh** command, it generated a free mesh on them instead. Now, you can use the new **Store Free Mesh if Mapped Mesh Fails** option to control whether the software retains that free mesh. For example, you may want to clear the **Store Free Mesh if Mapped Mesh Fails** check box when you want to see clearly any faces on which the software failed to generate a mapped mesh.

A new customer default lets you control the default setting for the **Store Free Mesh if Mapped Mesh Fails** option. To access this default:

1. From the **File** menu, choose **Utilities→Customer Defaults**.
2. In the **Customer Defaults** dialog box, choose **Simulation→Meshing**.
3. Click the **2D Mapped Meshing** tab and either select or clear the **Keep Free Meshes** check box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation→2D Mapped Mesh** 

Improved selection of nodes and elements within a solid mesh

What is it?

When you select entities within a solid mesh, you can now control whether the software includes any internal nodes and elements in your selection. The **Node and Element Display** dialog box has a new **Include Internal Nodes and Elements in Selection** option that lets you include nodes and elements in the interior of a solid mesh in your selection regardless of whether those internal nodes and elements are currently displayed. In previous releases, you could only select internal nodes and elements if the **Display Internal Edges** option in the **Mesh Display** dialog box was selected.

The **Include Internal Nodes and Elements in Selection** option works with all selection operations, including area selection, smart selection, and select all. By default, this option is turned on.

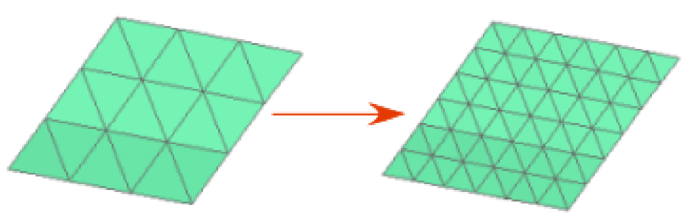
Where do I find it?

Application	Advanced Simulation, Design Simulation
Prerequisite	An active FEM file.
Menu	**Preferences→Node and Element Display**

Split Shell now splits a single triangle into four

What is it?

You can use the new **Triangle to 4 Triangles** option in the **Type** menu of the **Split Shell** dialog box to divide a single triangular element into four triangular elements. You can use **Triangle to 4 Triangles** when you need to locally refine a region within a triangular mesh.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Element Operations→Split Shell** 
Menu	**Edit→Element→Split Shell**

Material orientation

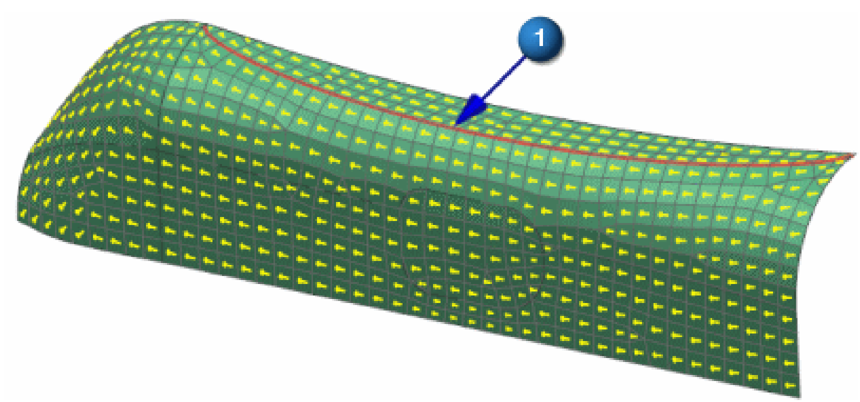
New material orientation definition methods

What is it?

New methods have been added for defining material orientation for shell and solid elements in the **Mesh Associated Data** dialog box, for the Nastran and Abaqus solvers.

- **Tangent Curve** (Shell or solid elements) — Lets you define the material orientation by selecting a curve or edge on your geometry. For each element, the software finds the closest point on the selected curve or edge to the element centroid and sets the primary direction of the material coordinate system as tangent to that curve or edge.

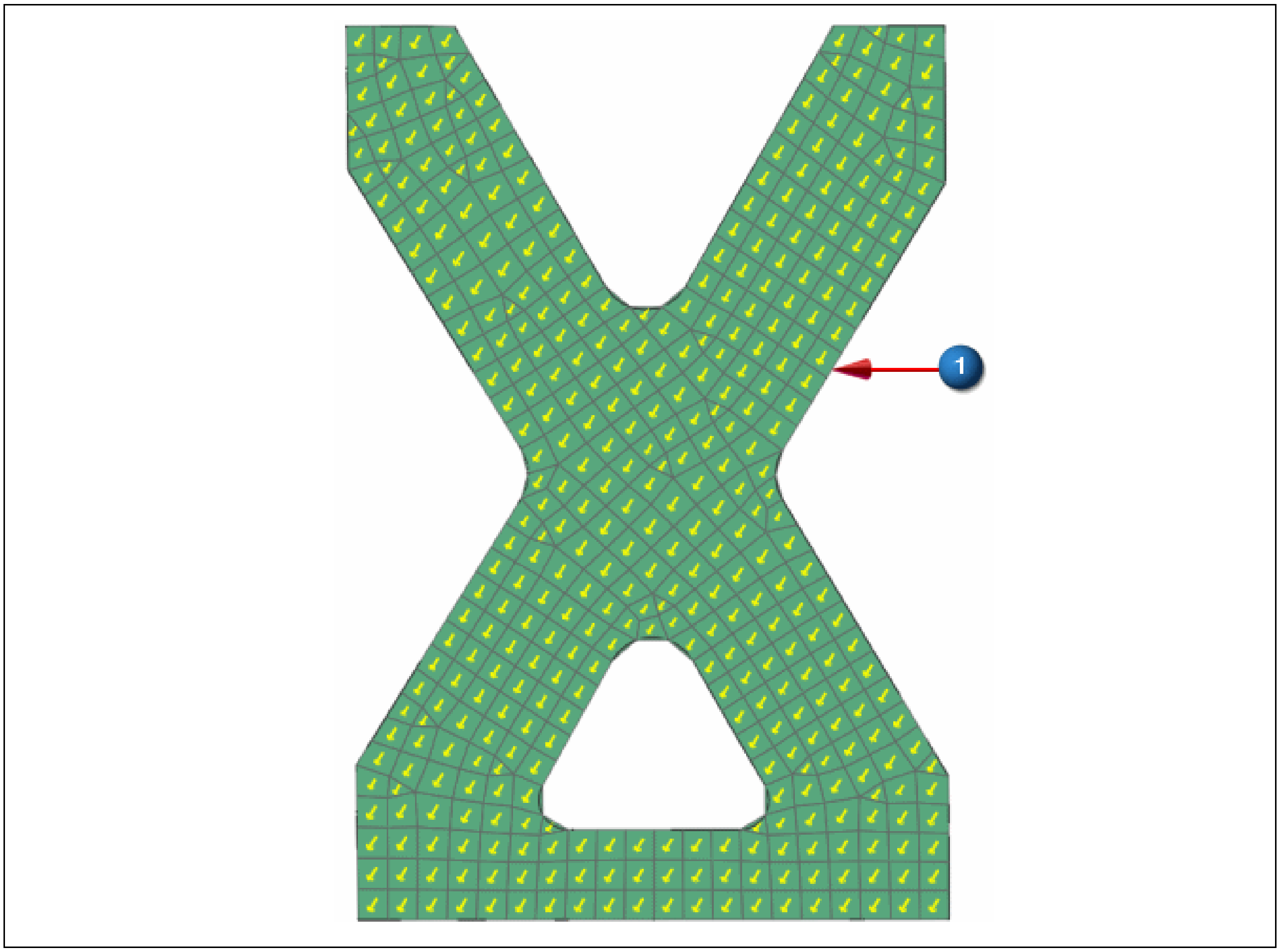
  In the following example, the material orientation represented by the yellow arrows was defined by selecting the edge  .

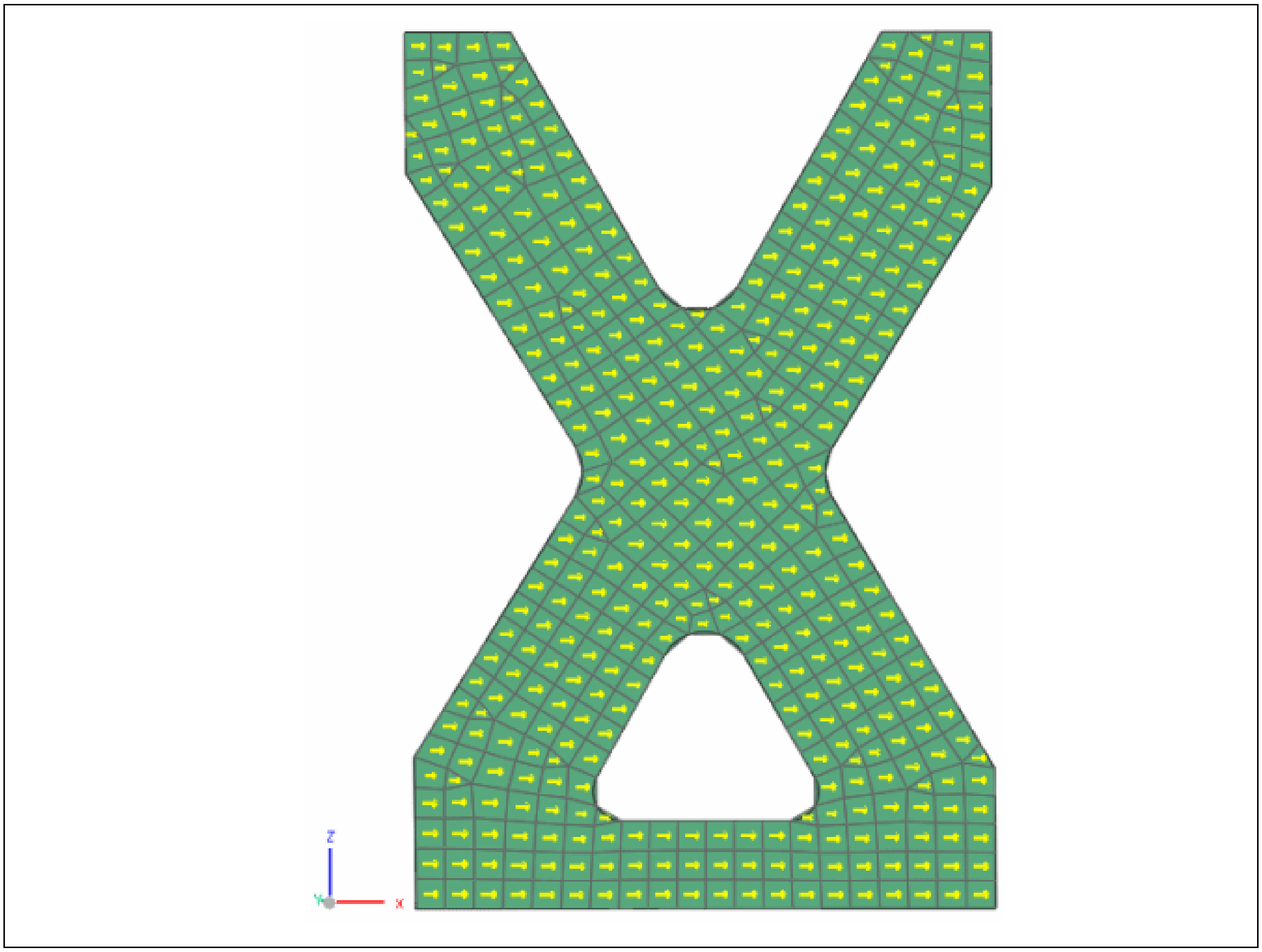
  #### Note

  When you define the material orientation for shell elements, you orient only the X-axis and Y-axis in the plane of the element. The Z-axis is fixed and is normal to the element. The orthogonality of X, Y, and Z means you only need to orient the X-axis. The right-hand rule applies. For solid elements, orienting two axes automatically derives the third.

- **Vector** (Shell or solid elements) — Lets you define the material orientation vector either by selecting an edge in the geometry to infer a vector (see first example below) or by using the NX vector tools to define the vector (see second example below).

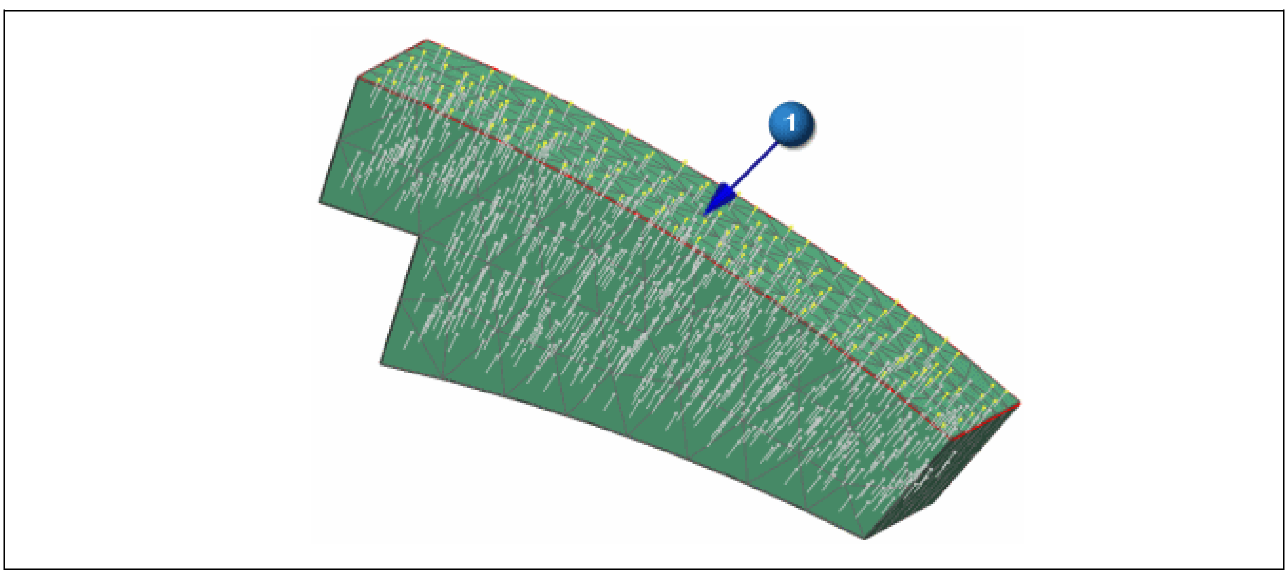


Vector inferred from edge (1)



Vector specified as X direction

- **Surface Normal** (Solid elements only) — Lets you define the material orientation by selecting a surface on the geometry. For each element, the software finds the closest point on the face to the element centroid and then sets the primary direction as normal to the face at that point.



Primary direction normal to selected face (1)

Note

You can view and control the display of the material orientation arrows in the graphics window using the **Finite Element Model Check** command. For more information, see **Unsatisfied xref title**.

Where do I find it?

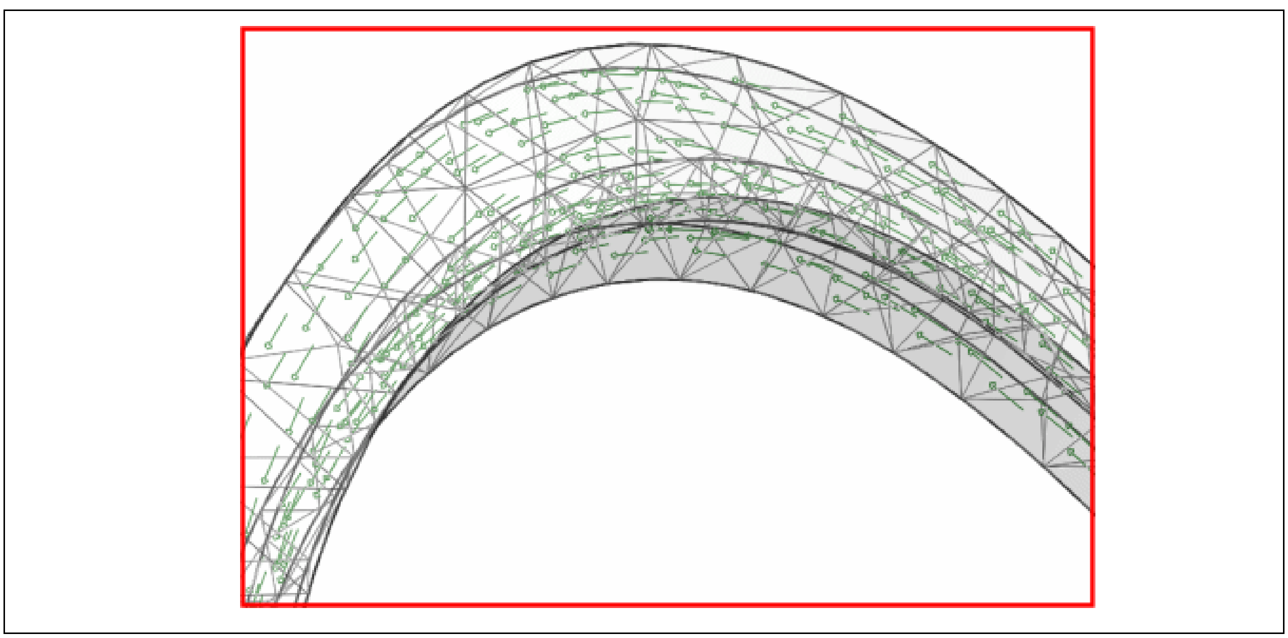
Application	Advanced Simulation
Prerequisite	An active FEM file.
Toolbar	**Advanced Simulation** toolbar→**Mesh Associated Data** 
Menu	**Tools**→**Mesh Associated Data**

View options for material orientation on solid elements

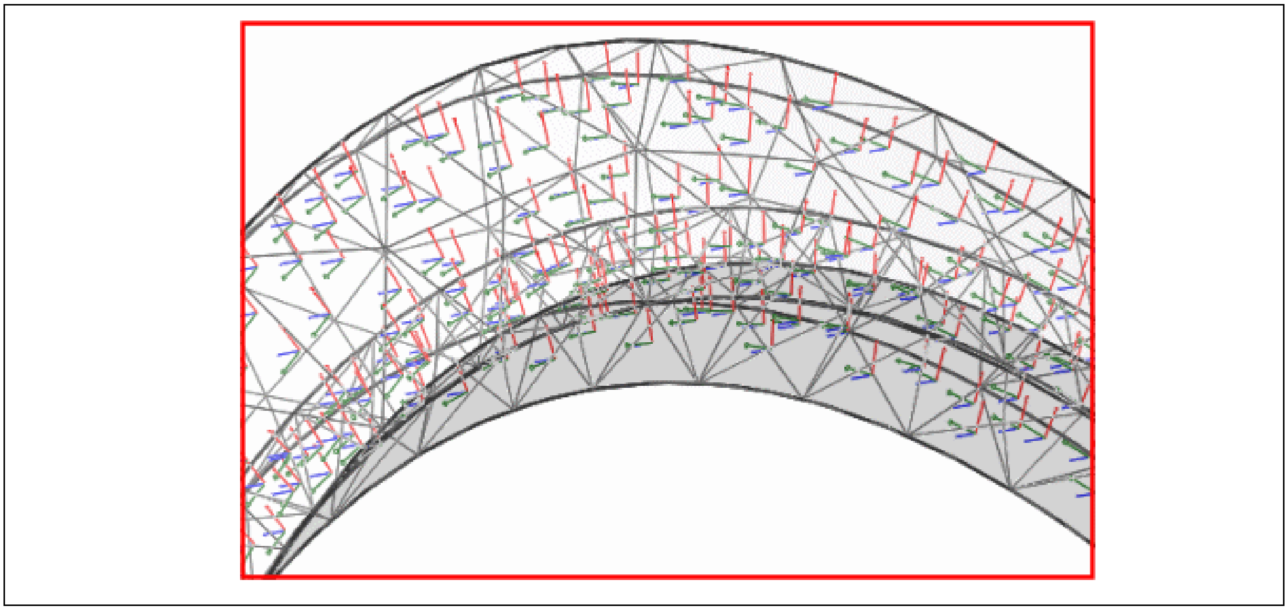
What is it?

A new option, **3D Element Material Orientation**, has been added to the **Model Check** dialog box that lets you view the material orientation display in the graphics window for solid elements.

After defining material orientation, you can view the material coordinate system for each element in the graphics window. You can view the first, second, and third directions of the material coordinate systems independently, or all three simultaneously. You can also choose the colors in which to display the directions of the material coordinate systems.

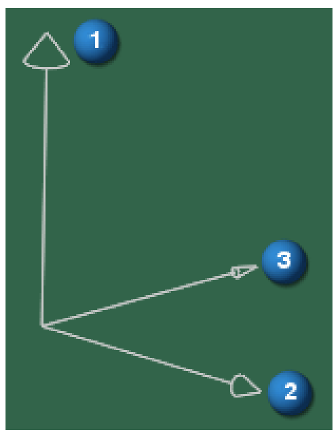


Only first direction shown (first direction is tangent to the curved geometry)



Complete coordinate systems shown (green is first direction; red is second direction)

In addition to using color, you can identify the first, second, and third directions in a material coordinate system by looking at the size of the arrow heads. In the following graphic, the first, second, and third directions are identified.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains solid elements with material orientation defined.
Toolbar	**Advanced Simulation** toolbar→**Finite Element Model Check**
Menu	**Analysis**→**Finite Element Model Check**

Assembly FEM

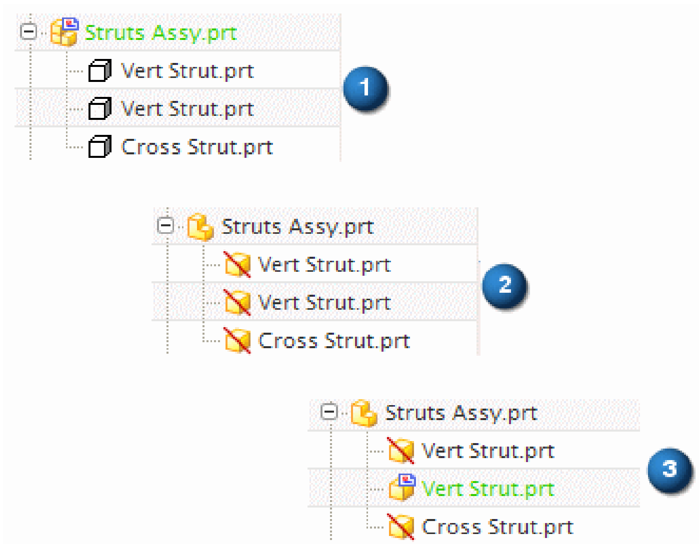
Map to individual components in a subassembly

What is it?

The **Unmap** command has been renamed to **Map at Next Level**.

Within an assembly FEM, you can use the **Map at Next Level** command to associate a component FEM with an individual component in a CAD subassembly without associating it with the entire subassembly.

This command changes the status of the CAD subassembly to **Not Mapped** and the status of the components to **Ignored** . Then you can right-click an individual component and select **Map New** or **Map Existing** to associate the component with a component FEM.



(1) Mapped subassembly; (2) Same subassembly after Map at Next Level command; (3) Individual component mapped to a FEM

You can also use the **Map Sub Assemblies at Next Level** command to perform the same function on the top-level CAD assembly. This command changes the status of all CAD subassemblies to **Not Mapped** and the status of all CAD components to **Ignored**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active assembly FEM with the assembly loaded
Simulation Navigator	Right-click a CAD assembly node→**Map at Next Level**
	Right-click the top-level CAD assembly node→**Map Sub Assemblies at Next Level**

Usability improvements to Assembly Label Manager

What is it?

When defining an assembly FEM, you can specify offsets for the identifier labels the software generates for nodes, elements, and coordinate systems across component FEMs. Several usability enhancements have been made to the **Assembly Label Manager** dialog box to make the process of specifying these offsets easier.

There are three methods for specifying how the automatic label offsets are applied:

- **Same Offsets in Component** — One value is used for **Offset to Nearest**. For each component, the offset is the same for nodes, elements, and coordinate systems. The software calculates the highest-needed offset and applies it uniformly to all offsets within the component. The nodes, elements, and coordinate systems start with the same ID number in each component. This is a new type.
- **Same Nearest Value for All** — One value is used for **Offset to Nearest**. For each component, the offset is calculated independently for nodes, elements, and coordinate systems.
- **Separate Nearest Value for All** — Lets you specify different **Offset to Nearest** values for nodes, elements, and coordinate systems. For each component, the offset is calculated independently for nodes, elements, and coordinate systems.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active assembly FEM
Simulation Navigator	Right-click the assembly FEM node and choose **Assembly Label Manager**.

Assembly FEM model checks improved

What is it?

Features have been added to the **Finite Element Model Check** dialog box specifically for Assembly FEMs.

Model Setup check

With an assembly FEM loaded and you open the **Model Check** dialog box, the following new **Model Setup** options are available.

Assembly Check

- **Label** — Checks for conflicts in node, element, and coordinate system labels across component FEMs and reports the label range in which the conflict occurs.
- **Connectivity** — Checks for component FEMs that are not connected.
- **Comprehensive** — Includes both the **Label** and **Connectivity** checks.

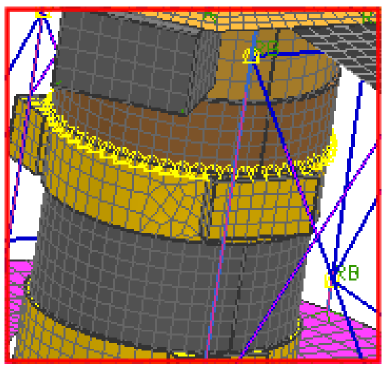
Assembly Load

- **All Components** — Includes all components associated with the assembly FEM in the model check, regardless of load status.
- **Fully Loaded Components** — Includes only fully loaded components in the model check.

Duplicate Nodes check and merge

The **Duplicate Nodes** check has been added for assembly FEMs. This command works as it does in component FEMs. However, it processes only the nodes that were created in the active assembly FEM and can merge a node created in the assembly FEM with a node in a component FEM.

- **Show Duplicate Nodes** — Displays the duplicate nodes in the active assembly FEM. The duplicate nodes are displayed in yellow.
- **Merge Duplicate Nodes** — Merges parent nodes in the active assembly FEM with any child nodes in the active assembly FEM or in a component FEM.



Duplicate nodes indicated with yellow circles

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active assembly FEM
Toolbar	**Advanced Simulation→Finite Element Model Check** 
Menu	**Analysis→Finite Element Model Check**

Move Component command replaces Reposition Component

What is it?

You can change the position of a non-associative component FEM within an assembly FEM. For consistency with the Assemblies application, the **Reposition Component** command has been replaced with the standard **Move Component** command.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active assembly FEM
Simulation Navigator	Right-click a component FEM→**Move Component**

Nastran environment

Updates to Strategy Parameters and Contact Parameters modeling objects

What is it?

This release includes updates to the NX Nastran **Strategy Parameters** and **Contact Parameters** modeling objects.

Strategy Parameters updates

In the **Strategy Parameters** dialog box:

- On the **Analysis Options** tab, a new **Automatic** option has been added to the **Stiffness Matrix Stabilization Factor (MSTAB)** list. With the **Automatic** option, the software automatically uses matrix stabilization if the ratio of maximum/minimum diagonal of factorized matrix is greater than 1.0E10.
- On the **Contact Control** tab, a new **Tensile Contact Forces (TNSLCF)** option has been added. Use this option to indicate whether tensile consistent contact forces on quadratic 3D contact segments are allowed.

See *NXSTRAT* in the *NX Nastran Quick Reference Guide* for more information.

Contact Parameters updates

In the **Contact Parameters** dialog box:

- Use the new **Include Shell Element Z-Offset** option to control whether the software includes any shell element z-offset when determining the contact surfaces. This option corresponds to the ZOFFSET field in the BCTPARM Bulk Data entry.
- Use the new **All Contact Elements Can Become Inactive** option to control whether all the contact elements can become inactive. This option corresponds to the CSTRAT field in the BCTPARM Bulk Data entry.

See *BCTPARM* in the *NX Nastran Quick Reference Guide* for more information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran as the specified solver

Toolbar	**Advanced Simulation→Modeling Objects** 
Menu	**Insert→Modeling Objects**

New hyperelastic material models

What is it?

This release includes support for new NX Nastran hyperelastic material models. Hyperelastic materials let you model materials that are nearly incompressible but which undergo large strains.

Hyperelastic Material Model	Corresponding ANSYS Command
Arruda-Boyce	MATHE bulk data entry with Model field = Aboyce
Foam	MATHE bulk data entry with Model field = Foam
Mooney-Rivlin	MATHE bulk data entry with Model field = Mooney
Ogden	MATHE bulk data entry with Model field = Ogden

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM, idealized part, or part with NX Nastran as the specified solver
Toolbar	**Advanced Simulation→Material Properties** 

Changes to the handling of MATS1 data

What is it?

When you export or solve a Nastran input file, Advanced Simulation now handles any defined material stress-strain (MATS1) data differently than in previous releases. In Advanced Simulation, you use the options in the **Stress-Strain Related Parameters** group in the **Isotropic Material** dialog box to define the stress-strain data.

These changes are:

- Depending on the option you select from the **Yield Function Criterion (YF)** list, you must now specify either a value in the **Initial Yield Point (LIMIT1)** or the **Initial Friction Angle (LIMIT2)** box.
  - If you choose **von Mises** or **Tresca**, you must specify an **Initial Yield Point (LIMIT1)** value. In Nastran, LIMIT1 should contain the material's Yield Strength (Y_S) value.

  - If you choose **Mohr-Coulomb** or **Drucker-Prager**, you must specify an **Initial Yield Point (LIMIT1)** value and an **Initial Friction Angle (LIMIT2)** value.

  Importantly, none of the materials in the NX material library currently have the LIMIT1 value defined. If you need to use a library material in a Nastran analysis, you must make a copy of that material and then edit the material to define the **Initial Yield Point (LIMIT1)** value.

- Advanced Simulation no longer corrects defined stress-strain curves to meet Nastran requirements. Previously, the software verified that your stress-strain curve had its first point at (0,0) and its second point at (Y_S/E, Y_S). If the data did not meet those requirements, the software fixed it automatically. Now, Advanced Simulation leaves the stress-strain data as you defined it. If the stress-strain data is not defined correctly when you solve your model, Nastran issues a fatal error.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM, idealized part, or part with NX Nastran as the specified solver
Toolbar	**Advanced Simulation→Material Properties**

Bolt pre-loads now supported for Response Simulation analyses

What is it?

You can use the **Bolt Pre-Load** command to to apply a pre-load to a bolt modeled with CBAR or CBEAM type beam elements in a Response Simulation (**SEMODES 103 - Response Simulation**) analysis. Bolt pre-loads are only supported in the first two subcases in a Response Simulation:

- Static Offset
- Stress Stiffening

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran as the specified solver and **SEMODES 103 - Response Simulation** as the solution type and with either **Subcase - Static Offset** or **Subcase - Stress Stiffening** active in the **Simulation Navigator**
Toolbar	**Advanced Simulation→Bolt Pre-Load**

New weld-like glue algorithm

What is it?

Advanced Simulation now supports the new weld-like glue algorithm that was introduced in the NX Nastran 6.1 release. A surface-to-surface glue condition on non-coincident shell or solid faces can introduce artificial rotational energy into the solution. Generally, the problem occurs because the spring-like glue elements do not transfer moments at the glue interface when glued faces are non-coincident and/or when loads are not normal to the glued faces. This is particularly noticeable in a normal mode solution when modes are found that contain an artificial rotational energy due to the glue condition. The new NX Nastran algorithm eliminates this artificial rotational energy.

In NX Nastran 6.1, two new fields were added to the BGPARM bulk data entry to allow you to specify and work with this new algorithm. In Advanced Simulation, the **Glue Parameters** modeling object has been updated to support the new BGPARM fields. In the **Glue Parameters** dialog box:

- The **Alternate Glue Formulation** option corresponds to the new GLUETYPE field on the BGPARM bulk data entry. Use this option to control the glue algorithm that NX Nastran uses.
  - Select **Weld-Like Connection** to use the new weld-like glue algorithm.
  - Select **Normal and Tangential Springs** to use the original, spring-like glue algorithm.
- The **Unitless Scale Factor for the Stiffness** option corresponds to the new PENGLUE field on the BGPARM bulk data entry. Use this option to specify the penalty factor for the weld-like glue algorithm.

> **Note**
>
> With the **Normal and Tangential Springs** option, use the **Penalty Normal Direction** and **Penalty Tangential Direction** options to specify the penalty factor.

In most cases, the new weld-like glue algorithm represents the connection stiffness more accurately than the spring-like glue algorithm because it transfers moments at the glue interface.

For more information, see *New Weld-Like Glue Method* in the *NX Nastran 6.1 Release Guide* and *BGPARM* 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 as the specified solver.
Toolbar	**Advanced Simulation→Modeling Objects** 
Menu	**Insert→Modeling Objects**

PBUSH structural damping enhancements

What is it?

For NX Nastran models, new options have been added to the Damping tab in the **PBUSH** physical property table dialog box. Use these options to define structural damping for each degree-of-freedom for CBUSH elements. The ability to define direction-dependent structural damping for the PBUSH bulk data entry was introduced in the NX Nastran 6.1 release. Previously, you could only define a single, structural damping value (GE1) that applied to all 6 degrees-of-freedom.

The new options in the **PBUSH** dialog box let you specify separate structural damping values for the X, Y, and Z translations and rotations. The values you specify correspond to the GE1, GE2, GE3, GE4, GE5, and GE6 fields in the PBUSH bulk data entry. If you define a value for any of the **Structural** options on the Damping tab, you should define a value for all degrees-of-freedom that are critical to the result, because a blank field defaults to a value of zero.

This release also includes support for the new BSHDAMP parameter, which lets you optionally ignore any of the GE2-GE6 fields and only use the GE1 field. By default, the software considers the new GE2-GE6 fields. The new **BSHDAMP** list in the **Solution Parameters** dialog box (available from the **Modeling Objects Manager** dialog box) lets you control the setting for this parameter:

- If you select **SAME** (`PARAM, BSHDAMP=SAME`), the software ignores the GE2-GE6 fields and only considers the single, structural damping value defined in the GE1 field. This corresponds to the behavior available in previous releases.
- If you select **DIFF** (`PARAM, BSHDAMP=DIFF`), the software considers the GE2-GE6 fields.

For more information, see the *NX Nastran 6.1 Release Guide* and *PBUSH* in the *NX Nastran Quick Reference Guide*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file active with NX Nastran as the specified solver
Toolbar	**Advanced Simulation→Physical Properties** 
Menu	**Insert→Physical Properties**

PSHELL structural damping enhancements

What is it?

Beginning with the NX Nastran 6.1 release, the PSHELL bulk data entry supports including the structural damping coefficient (GE) on the associated materials:

- MID1 = membrane material
- MID2 = bending material
- MID3 = transverse shear
- MID4 = membrane-bending coupling

Previously, NX Nastran used the structural damping coefficient defined on the MID1 material for all materials associated with the PSHELL entry.

Advanced Simulation now supports the new NX Nastran SHLDAMP parameter. You use the SHDAMP parameter to turn the structural damping coefficient capability on and off for a given analysis. The new **SHLDAMP** list in the **Solution Parameters** dialog box (available from the **Modeling Objects Manager** dialog box) lets you set this parameter.

- If you select **SAME** (`PARAM, SHLDAMP=SAME`), which is the default, the software uses the structural damping coefficient (GE) defined on the MID1 material for the PSHELL entry for all MIDi materials for that PSHELL.
- If you select **DIFF** (`PARAM, SHLDAMP=DIFF`), the software uses the structural damping coefficient (GE) defined on each MIDi for the PSHELL entry, provided that the GE field is defined for any of the MID2, MID3, and/or MID4 materials in at least one PSHELL entry in the input file. With this option, if any structural damping coefficient (GE) value is blank, NX Nastran treats it as having a value of zero.

For more information, see the *PSHELL Structural Damping* in the *NX Nastran 6.1 Release Guide* and *SHLDAMP* in the *NX Nastran Quick Reference Guide*.

Where do I find it?

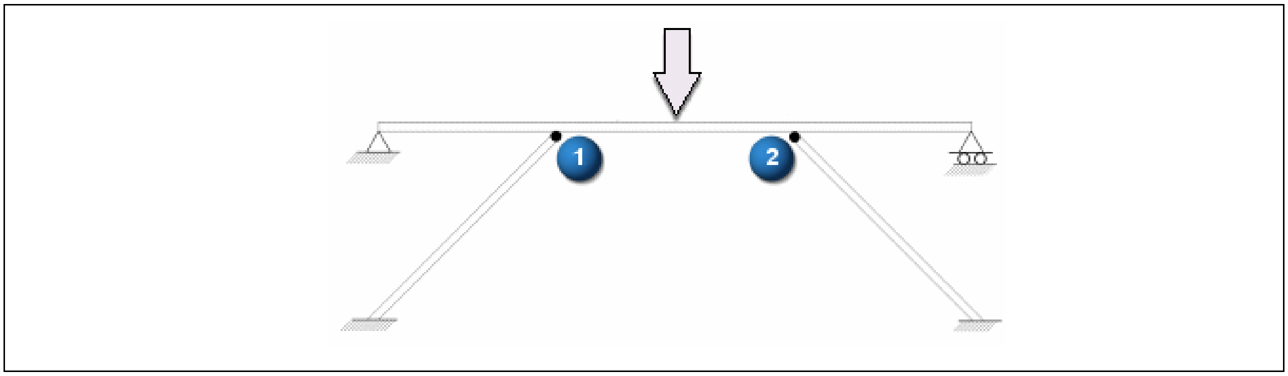
Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran as the specified solver.
Toolbar	**Advanced Simulation→Modeling Objects**
Menu	**Insert→Modeling Objects**

End releases for beam and bar elements

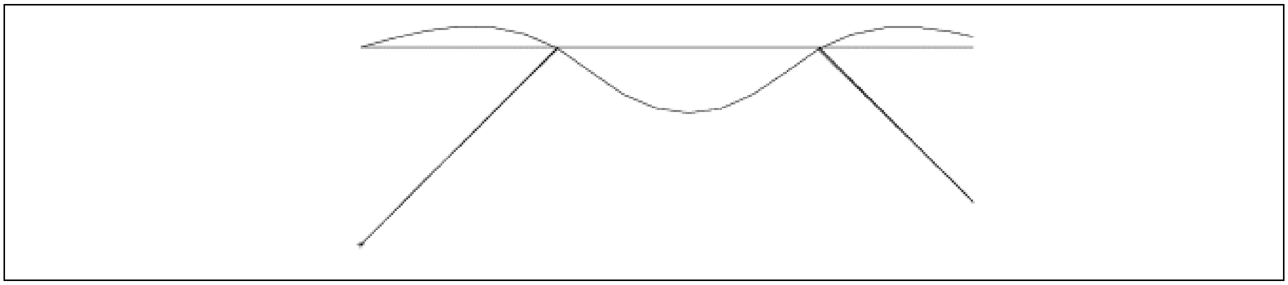
What is it?

You can now define end releases (also known as pin flags) at either end of a Nastran CBEAM or CBAR element in the **Element Associated Data** dialog box. Previously, you could define pin flags only at the **Mesh Associated Data** level.

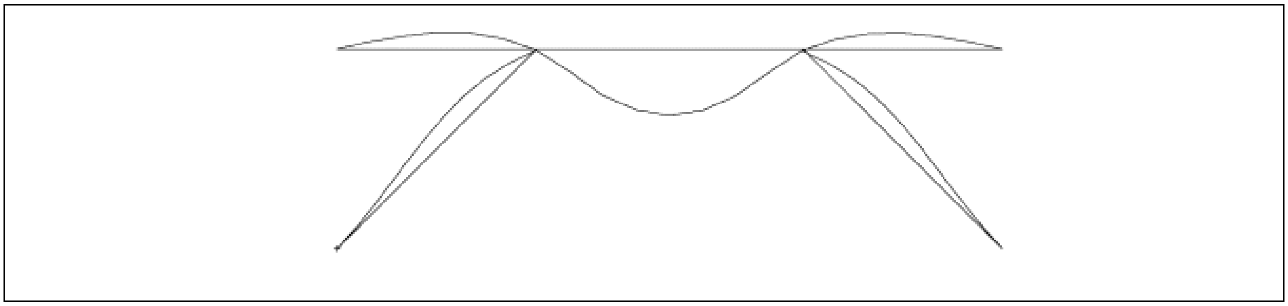
In a beam, end releases remove connections between a node and selected degrees of freedom. The degrees of freedom are defined in the element coordinate system. For the following example, a force is defined on the horizontal span. Because end releases are defined at the ends of the bracing beams, the solver does not transfer the moment load from the horizontal beam to the bracing beams. The moments are transferred across the other elements in the horizontal span.



Bridge model. (1) and (2) represent the end releases; force applied to horizontal span



Bridge model; deflected shape with end releases



Bridge model; deflected shape without end releases

In the **Element Associated Data** dialog box, under the **End Releases** group (for End A and or End B of the element), you can set **DOF1–6** to **On** to disconnect the following forces:

- DOF1 — axial force in Plane 1
- DOF2 — shearing force in Plane 1
- DOF3 — shearing force in Plane 2
- DOF4 — axial torque in Plane 2
- DOF5 — moment in Plane 2
- DOF6 — moment in Plane 1

Note

Plane 1 is the XY plane formed by the X and Y axes in the element coordinate system. Plane 2 is the XZ plane formed by the X and Z axes.

In the previous example, **DOF6** is set to **On** at the ends of the bracing beams (the model is shown in the XY plane).

For more information, see Element associated data overview in the Advanced Simulation help.

Why should I use it?

You can define a release at the ends of beam or bar elements to model hinged or pinned connections.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains CBEAM or CBAR elements.
Toolbar	**Element Operations→Element Associated Data**
Menu	**Edit→Element→Modify Associated Data**

Optional torsional mass moments of inertia for CROD and CBAR elements

What is it?

Advanced Simulation now supports the new TORSIN parameter that was introduced in the NX Nastran 6.1 release. The TORSIN parameter lets you optionally include the torsional mass moment of inertia in the CROD and CBAR element mass matrices. By default, NX Nastran does not calculate torsional mass for CROD or CBAR elements, though it does for CBEAM elements. This can lead to differences when you compare results between equivalent models.

The new **TORSIN** box in the **Solution Parameters** dialog box (available from the **Modeling Objects Manager** dialog box) lets you control the setting for this parameter:

- If you enter 0, the software does not include the torsional mass moment of inertia for CROD and CBAR elements (the default).
- If you enter 1, the software includes the torsional mass moment of inertia for CROD and CBAR elements.
- If you enter 2, the software includes the torsional mass moment of inertia for CBAR elements only.
- If you enter 3, the software includes the torsional mass moment of inertia for CROD elements only.

For more information, see the *NX Nastran 6.1 Release Guide* and *PBUSH* in the *NX Nastran Quick Reference Guide*.

CBAR axial torsional mass moment of inertia calculation

NX Nastran calculates the CBAR axial torsional mass moment of inertia similarly to the CBEAM element using the equation:

$I_{xx} = \rho L(I_1+I_2)$

where:

I_{xx} = torsional mass moment of inertia

ρ = density

L = the length of the element

I_1 and I_2 = area moments of inertia

CROD axial torsional mass moment of inertia calculation

NX Nastran calculates the CROD axial torsional mass moment of inertia using the equation:

$I_{xx} = \rho L I_x$

where I_x = J = torsional constant.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran as the specified solver
Toolbar	**Advanced Simulation→Modeling Objects** 
Menu	**Insert→Modeling Objects**

Specifying additional keywords and system cells

What is it?

A new **Additional Keywords** option has been added to the **nastran Command Keywords** options on the Solve Options page of the **Solver Parameters** dialog box. The **nastran Command Keywords** options include a subset of frequently used keywords for the nastran command, such as **sdirectory** and **old**. You can use the new **Additional Keywords** option to manually specify any additional keywords for the nastran command or any system cells for the NASTRAN executive control statement. The ability to manually specify specific keywords or system cells gives you greater control over how Nastran solves your model.

For more information, see *the nastran command* and *the NASTRAN statement* in the *NX Nastran Quick Reference Guide* for a complete list of the supported keywords and system cells.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran or MSC Nastran as the specified solver
Simulation Navigator	Right-click the Simulation→**New Solution**, or right-click the current solution→**Edit Solution**
Menu	**Insert→Solution**

Results support enhancements

Support for nonlinear stress results output

What is it?

Use the options on the new **Nonlinear Stress** tab in the **Structural Output Requests** dialog box to request the output of nonlinear element stresses for SOL 106 (NLSTATIC 106) analyses. These options let you control how the software presents the tabular listing of results, the location where the results are output, as well as the elements for which the software calculates the stresses. When you create a **Nonlinear Stress** type of output request, the software creates an NLSTRESS Case Control entry in the Case Control section of your Nastran input file.

A new customer default lets you control whether the **Enable NLSTRESS Request** option on the **Nonlinear Stress** tab is selected by default. To set this default:

1. From the **File** menu, choose **Utilities→Customer Defaults**.
2. In the **Customer Defaults** dialog box, choose **Simulation→NASTRAN**.
3. Click the **Solution** tab and either select or clear the **Nonlinear Stress** option.

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** 
Menu	**Insert→Modeling Objects**

Shell thickness output for advanced nonlinear analyses

What is it?

Use the options on the new **Shell Thickness** tab in the **Structural Output Requests** dialog box to request the output of shell element thickness values for SOL 601 and 701 (ADVNL 601 and 701) analyses. NX Nastran only outputs shell thickness results for large strain analyses (analyses in which you include the parameter PARAM,LGSTRN,1). To set this parameter, select the new **Large Strains** check box on the Parameters page of the **Create Solution** or **Edit Solution** dialog box.

When you create a **Shell Thickness** type of output request, the software creates a SHELLTHK Case Control entry in the Case Control section of your Nastran input file. For more information on shell thickness results, see *SHELLTHK* in the *NX Nastran Quick Reference Guide*.

A new customer default lets you control whether the **Enable SHELLTHK Request** option on the **Nonlinear Stress** tab is selected by default. To set this default:

1. From the **File** menu, choose **Utilities→Customer Defaults**.
2. In the **Customer Defaults** dialog box, choose **Simulation→NASTRAN**.
3. Click the **Solution** tab and either select or clear the **Shell Thickness** option.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran as the specified solver and ADVNL 601,106, ADVNL 601, 129, or ADVNL 701 as the solution type
Simulation Navigator	Either right-click a Simulation and select **Create Solution**, or right-click the appropriate solution and select **Edit Solution**

Support for gasket results output

What is it?

Use the options on the new **Gasket Result** tab in the **Structural Output Requests** dialog box to request the output of gasket results for SOL 601 (ADVNL 601,106 and ADVNL 601,129) analyses. Gasket results include gasket pressure, gasket closure, plastic gasket closure, gasket yield stress, and gasket status. When you create a **Gasket Results** type of output request, the software creates an GKRESULTS Case Control entry in the Case Control section of your Nastran input file.

A new customer default lets you control whether the **Enable GKRESULTS Request** option on the **Gasket Results** tab is selected by default. To set this default:

1. From the **File** menu, choose **Utilities→Customer Defaults**.
2. In the **Customer Defaults** dialog box, choose **Simulation→Nastran**.
3. Click the **Solution** tab and either select or clear the **Gasket Results** option.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran as the specified solver and ADVNL 601,106 and ADVNL 601,129 as the solution type
Toolbar	**Advanced Simulation→Modeling Objects** 
Menu	**Insert→Modeling Objects**

Import and export enhancements

Nastran import and export support enhancements

What is it?

This release includes enhancements to the import and export support for Nastran Bulk Data entries and Case Control commands.

Newly supported bulk data entries and case control commands

Name	NX Nastran Import/Export Support	MSC Nastran Import/Export Support	Notes
CDAMP3 and CDAMP4	Yes (import support only)	Yes (import support only)	See **Unsatisfied xref title** for more information.
CELAS3 and CELAS4	Yes (import support only)	Yes (import support only)	See **Unsatisfied xref title** for more information.
CMASS3 and CMASS4	Yes (import support only)	Yes (import support only)	See **Unsatisfied xref title** for more information.
CPYRAM	Yes	No	See **Unsatisfied xref title** for more information.
GKRESULTS (Case Control command)	Yes	No	See **Unsatisfied xref title** for more information.
MATG	Yes	Yes	See **Unsatisfied xref title** for more information.
MATHE	Yes	No	See **Unsatisfied xref title** for more information.
MATS1	Yes	Yes	See **Unsatisfied xref title** for more information.
MEFFMASS (Case Control command)	Yes	Yes	MEFFMASS is now supported for import into Advanced Simulation.
NLSTRESS (Case Control command)	Yes	Yes	See **Unsatisfied xref title** for more information.
SHELLTHK (Case Control command)	Yes	No	See **Unsatisfied xref title** for more information.
SUPORT	Yes	No	See **Unsatisfied xref title** for more information.
USET	Yes	Yes	See **Unsatisfied xref title** for more information.
USET1	Yes	Yes	See **Unsatisfied xref title** for more information.

Enhancements for previously supported bulk data entries and case control commands

Name	NX Nastran Import/Export	MSC Nastran	Newly Supported Fields and Other	Notes

	Support	Import/Export Support	Enhancements	
BCRESULTS (Case Control command)	Yes	No	SET, SEPDIS	
BGPARM	Yes	No	GLUETYPE, PENGLUE	See **Unsatisfied xref title** for more information.
BLSEG	Yes	Yes	For export, the software now supports the use of both increasing and decreasing THRU options.	
CBAR	Yes	Yes	PA, PB	Pin flags are now supported as **Element Associated Data**. See **Unsatisfied xref title** for more information.
MAT1	Yes	Yes	GE	• The MCSID field is not yet supported. • See **Unsatisfied xref title** for more information.
MAT2	Yes	Yes	GE	• The MCSID field is not yet supported. • See **Unsatisfied xref title** for more information.
MAT3	Yes	Yes	GE	See **Unsatisfied xref title** for more information.
MAT8	Yes	Yes	GE	See **Unsatisfied xref title** for more information.
MAT9	Yes	Yes	GE	See **Unsatisfied xref title** for more

				information.
PBUSH	Yes	Yes	GE2–GE6	This release adds support for the GE2-E6 fields for NX Nastran. These fields were previously supported for MSC Nastran. See **Unsatisfied xref title** for more information.
PLOADX1	Yes	Yes		For export, the software now supports time-dependent pressure loads.
TABLES1	Yes	Yes		You can now define TABLES1 data in the context of MATS1 material data. Previously, you could only define TABLES1 data in the context of a MATHP type material. See **Unsatisfied xref title** for more information.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

Nastran parameter support updates

What is it?

Support has been added for a number of new NX Nastran and MSC Nastran parameters, and the default values of several existing parameters have been changed. When you create a **Solution Parameters** type of **Modeling Object**, these newly supported parameters now appear in the **Solution Parameters** dialog box.

For more information on MSC Nastran parameters, see the *MSC Nastran Quick Reference Guide*.

Newly supported parameters

Name	NX Nastran Import/Export Support	MSC Nastran Import/Export Support	Notes
ALPHA1FL	N/A	Yes	
ALPHA2FL	N/A	Yes	
ARBMASP	N/A	Yes	
ARBMFEM	N/A	Yes	
ARBMPS	N/A	Yes	
ARBMSS	N/A	Yes	
ARBMSTYP	N/A	Yes	
AUTOGOUT	N/A	Yes	
BSHDAMP	Yes	N/A	See **Unsatisfied xref title** for more information.
CFDIAGP	N/A	Yes	
CFRANDEL	N/A	Yes	
CORITAN	N/A	Yes	
CQC	N/A	Yes	
DIROUT	N/A	Yes	
DREPONLY	Yes	N/A	
DV3PASS	N/A	Yes	
ENFMETH	N/A	Yes	
ESLFSAV	N/A	Yes	
FASTFR	N/A	Yes	
FULLSEDR	N/A	Yes	
HTOCITS	N/A	Yes	
HTOCPRT	N/A	Yes	
HTOCTOL	N/A	Yes	
MDOPT14	N/A	Yes	
MDOTM	N/A	Yes	
MDOTMFAC	N/A	Yes	
MDOF	Yes	N/A	
MGRID	Yes	N/A	

MHRED	N/A	Yes	
NEWMARK	N/A	Yes	
OSETELE	N/A	Yes	
OSETGRD	N/A	Yes	
PATPLUS	N/A	Yes	
PERCENT	N/A	Yes	
RADMOD	N/A	Yes	
RANCPLX	Yes	N/A	
ROTSYNC	Yes	N/A	
RSTTEMP	N/A	Yes	
SHLDAMP	Yes	Yes (introduced in NX 5)	See **Unsatisfied xref title** for more information.
SPARSEPH	N/A	Yes	
SQSETID	N/A	Yes	
SRCOMPS	Yes	Yes (introduced in NX 5)	SRCOMPS controls the computation and printout of ply strength ratios.
TCHECK	N/A	Yes	
TDMIN	N/A	Yes	
TFSTMFAC	N/A	Yes	
TORSIN	Yes	N/A	See **Unsatisfied xref title** for more information.
WRH	N/A	Yes	
ZROVEC	N/A	Yes	

NX Nastran Parameters with updated default values

Parameter	New Default Value	Previous Default Value
BOLTFACT	1.0E7	1.0E6
K6ROT	100.0	0.0 and 100
SUBID	1	0

MSC Nastran Parameters with updated default values

Parameter	New Default Value	Previous Default Value
DESPCH1	0	6
ERROR	-1	0

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran or MSC Nastran specified as the solver
Toolbar	**Advanced Simulation→Modeling Objects** 
Menu	**Insert→Modeling Objects**

Nastran system cell support enhancements

What is it?

This release includes enhanced support for Nastran system cells. System cells are Executive System operational parameters specified by the NASTRAN statement. These enhancements include:

- Import support for NX Nastran and MSC Nastran system cells. When you import a Nastran input file, Advanced Simulation now imports any system cell settings defined in that file. In previous releases, Advanced Simulation did not import any system cells specified in your Nastran input file.
- Support for new NX Nastran and MSC Nastran system cells, as well as updates to currently supported system cells. When you create a **System Cells** type of **Modeling Object**, these newly supported system cells now appear in the **System Cells** dialog box.

NX Nastran system cell support updates

System Cell	Name	Notes
206	DCMPSEQ	Default value = 4
210	None	
252	None	
357	RSEQCONT	
370	QRMETH	Acceptable values: 3 or 5
404	None	
413	OP2FMT	

415	OP4FMT	
416	INP4FMT	
424	SPCHOL	Acceptable values: 4 or 5
444	TEMPWARN	
461	None	
462	None	
463	None	
464	None	

For more information about these system cells, see the *NX Nastran Quick Reference Guide*.

MSC Nastran system cell support updates

System Cell	Name	Notes
205	None	
219	None	
220	None	
221	None	
274	DBCFACT	
311	TBCMAG	
408	DEF_DENS	
410	DEF_TECO	
411	DEF_TEIJ	
412	DEF_DAMP	
428	None	
431	NONLRGAP	The name of the system cell is NONLRGAP and not NLRGAP, as documented by MSC.
445	MNLQ4C	

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file with NX Nastran or MSC Nastran specified as the solver
Toolbar	**Advanced Simulation→Modeling Objects**
Menu	**Insert→Modeling Objects**

Extended mesh validity checking available during import

What is it?

A new **Enable extended data checking** option has been added to the **Advanced Options** group in the **Import Simulation** dialog box. When you import a Nastran .dat or .op2 file, select **Enable extended data checking** to have the software check for the existence of each node on a given element in your input file. If any of an element's nodes are missing from the input file, the software:

- Does not import the element.
- Issues an error message when it first encounters an element with missing nodes.
- Clearly lists that the element failed to import in the **Analysis of Import** report as shown below.

```
----------------------
  ANALYSIS OF IMPORT
----------------------
************************************************************
*   Card    |   Cards     |  Entities  *  Min id  *  Max id  *
*   Name    | Recognized  |  Imported  *          *          *
************************************************************
*  CORD2R   | 1           | 1          * 2        * 2        *
*  CQUAD4   | 1           | 0          *          *          *
*  GRID     | 3           | 3          * 1        * 3        *
*  MAT1     | 1           | 1          * 100      * 100      *
*  PARAM    | 2           | 2          *          *          *
*  PSHELL   | 1           | 1          * 1000     * 1000     *
************************************************************
```

In previous releases, if you imported a Nastran input file in which even a single node was missing, the import operation appeared to proceed normally, though the software did not import the mesh. The software did not issue any error messages in the import summary, and it created a mesh node in the **Simulation Navigator**. The only indication of a problem was the absence of a displayed model in the graphics window.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

Enhanced import support for connection elements

What is it?

You can now import the following additional types of Nastran CDAMP, CELEAS, and CMASS connection elements from Nastran .dat and .op2 files into Advanced Simulation:

- CDAMP3 and CDAMP4
- CELAS3 and CELAS4
- CMASS3 and CMASS4

Currently, these types of elements are not directly supported in Advanced Simulation. During the import process, the software converts these elements to the most closely related, supported element types, as shown in the following table:

Nastran Element Type	Type Converted to During Import
CDAMP3	CDAMP1
CDAMP4	CDAMP2
CELAS3	CELAS1
CELAS4	CELAS2
CMASS3	CMASS1
CMASS4	CMASS2

For more information on these types of elements, see the *NX Nastran Quick Reference Guide* and the *NX Nastran Element Library Reference Manual*.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

Import changes for SPOINT entries

What is it?

Nastran uses the SPOINT bulk data entry to define scalar points. Scalar points have a single degree-of-freedom. In Nastran:

- You can explicitly define a scalar point by creating an SPOINT entry in the bulk data section of your input file.
- You can implicitly define a scalar point by referencing the ID of a scalar point on a CDAMP1/2/3/4, CMASS1/2/3/4, or CELAS1/2/3/4 element entry. If you specify a scalar point on one of these entries, you do not need to include an SPOINT entry in the bulk data section of your input file.

Advanced Simulation now allows you to import implicitly defined SPOINT bulk data entries. If your .dat file contains CDAMP1/2/3/4, CMASS1/2/3/4, or CELAS1/2/3/4 elements that reference an SPOINT, the software imports that SPOINT as a node, places its location at the origin, and fixes DOF 23456. The software gives the new node the same ID as the original SPOINT, and it modifies any element definitions that originally referenced the SPOINT to reference the new node instead. If you later export this model back to Nastran, the software writes this node out using a Nastran GRID bulk data entry that is fixed in DOF 23456.

For more information on SPOINTs, see the *SPOINT* in the *NX Nastran Quick Reference Guide* and *Understanding Scalar Points* in the *NX Nastran User's Guide*.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

Import support for HAT1 beam cross sections

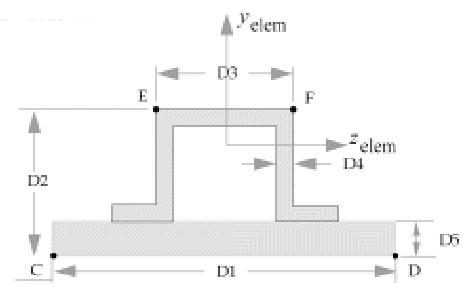
What is it?

For the PBARL and PBEAML beam cross section property bulk data entries, you can now import the Nastran HAT1 beam cross section type (TYPE field =HAT1). This allows you to import bar and beam elements that have a HAT1 type cross section defined in your input file. In previous release, only ROD, TUBE, I1, CHAN, BOX, and BAR type sections were supported.

> **Note**
>
> Because the **Section** dialog box in Advanced Simulation does not support HAT1 as a standard beam cross section shape, Advanced Simulation imports HAT1 cross sections as user-defined thin walled sections.

When you solve your model, the software computes the following properties for HAT1 cross sections:

- Area
- Iz, Iy, Iyz
- K
- Y_{elem} and Z_{elem}
- The X and Y components of stress recovery points C, D, E, and F



Definition of HAT1 beam cross section geometry and stress recovery points

For more information, see *PBARL* and *PBEAML* in the *NX Nastran Quick Reference Guide*.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

Additional import support for Response Simulation

What is it?

You can now import the following loads and boundary conditions used in SOL 103-Response Simulation analyses:

- **Fictitious Support** (SUPORT)
- **Enforced Motion Location** (USET,U2 and USET1,U2)
- **Nodal Force Location** (USET,U3 and USET1,U3)

For more information, see *SUPORT*, *USET*, and *USET1* in the *NX Nastran Quick Reference Guide*.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

Abaqus environment

Abaqus keyword support enhancements

What is it?

This release includes improved import and export support for the following Abaqus keywords:

Command	Supported Parameters	Import Support	Export Support	For More Information
*CLEARANCE		Yes	Yes	See **Unsatisfied xref title** for more information.
*GAP CONDUCTANCE		Yes	Yes	See **Unsatisfied xref title** for more information.
*GASKET BEHAVIOR	NAME	Yes	Yes	See **Unsatisfied xref title**
*GASKET ELASTICITY	COMPONENT, VARIABLE	Yes	Yes	See **Unsatisfied xref title**
*GASKET SECTION	ELSET, BEHAVIOR, MATERIAL, ORIENTATION, STABILIZATION STIFFNESS	Yes	Yes	See **Unsatisfied xref title**
*GASKET THICKNESS BEHAVIOR	DIRECTION, TENSILE STIFFNESS FACTOR, TYPE, VARIABLE, SLOPE DROP, YIELD ONSET	Yes	Yes	See **Unsatisfied xref title**
*HYPERELASTIC	ARRUDA-BOYCE, MARLOW, MOONEY-RIVLIN, NEO HOOKE, OGDEN, POLYNOMIAL, REDUCED POLYNOMIAL, TEST DATA, VAN DER WAALS, YEOH	No	Yes	See **Unsatisfied xref title**
*HYPERFOAM	TEST DATA	No	Yes	See **Unsatisfied xref title**
*INCLUDE	INPUT	Yes	No	See **Unsatisfied xref

				title**
*OUTPUT/*NODE OUTPUT/*ELEMENT OUTPUT		Yes	Yes	See **Unsatisfied xref title** for more information.

Enhancements for previously supported keywords

Name	Newly Supported Parameters	Notes
*ORIENTATION	NAME, SYSTEM	The *ORIENTATION keyword is now supported for import. Previously, *ORIENTATION was only supported for export. See **Unsatisfied xref title** for more information.
*PRE-TENSION SECTION	SURFACE	See **Unsatisfied xref title** for more information.

Support for new element types

This release also includes support for several new types of Abaqus elements. See **Unsatisfied xref title** for more information.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation** or **File→Export→Simulation**

New hyperelastic material models

What is it?

This release includes support for an expanded range of Abaqus hyperelastic material models, including several models in which you can use test data to define certain material properties. This release also includes support for a new **Gasket Behavior** material.

- Hyperelastic materials let you model materials that are nearly incompressible but which undergo large strains.
- The Abaqus **Gasket Behavior** material lets you model gasket behavior properties, such as elastic properties for the membrane and transverse shear behaviors of a gasket, for gasket analyses.

New materials

Material	Corresponding Keyword
Arruda-Boyce	*HYPERELASTIC keyword with both the ARRUDA-BOCYE parameter (constants option)
Arruda-Boyce Test Data	*HYPERELASTIC keyword with the ARRUDA-BOCYE and TEST DATA INPUT parameters
Foam	*HYPERFOAM keyword (constants option)
Foam Test Data	*HYPERFOAM keyword with the TEST DATA INPUT parameter
Gasket Behavior	*GASKET BEHAVIOR keyword
Marlow	*HYPERELASTIC keyword with the MARLOW parameter
Mooney-Rivlin	*HYPERELASTIC keyword with the MOONEY-RIVLIN parameter (constants option)
Neo Hooke	*HYPERELASTIC keyword with the NEO HOOKE parameter (constants option)
Neo Hooke Test Data	*HYPERELASTIC keyword with both the NEO HOOKE and TEST DATA INPUT parameters
Ogden	*HYPERELASTIC keyword with the OGDEN parameter (constants option)
Ogden Test Data	*HYPERELASTIC keyword with the OGDEN and TEST DATA INPUT parameters
Polynomial	*HYPERELASTIC keyword with the POLYNOMIAL parameter (constants option)
Reduced Polynomial	*HYPERELASTIC keyword with the REDUCED POLYNOMIAL parameter (constants option)
Van Der Waals	*HYPERELASTIC keyword with the VAN DER WAALS parameter
Yeoh	*HYPERELASTIC keyword with the YEOH parameter (constants option)
Yeoh Test Data	*HYPERELASTIC keyword with both the YEOH and TEST DATA INPUT parameters

Defining material constants using test data

With the test data material types, you can use a field to specify the deformation modes that define the material constants with test (experimental) data.

- Use the **UNIAXIAL Tension/Compression** option to specify uniaxial test data. This option corresponds to the Abaqus *UNIAXIAL TEST DATA keyword. With **UNIAXIAL Tension/Compression**, you define a table field in which you list the material's nominal stress (T_U) and nominal strain values (ε_U) on each line.
- Use the **BIAXIAL Tension** option to specify biaxial test data. This option corresponds to the Abaqus *BIAXIAL TEST DATA keyword. With **BIAXIAL Tension**, you define a table field in which you list the material's nominal stress (T_B) and nominal strain values (ε_B) on each line.
- Use the **PLANAR - Pure Shear** option to specify planar (or pure shear) data. This option corresponds to the Abaqus *PLANAR TEST DATA keyword. With **PLANAR - Pure Shear**, you define a table field in which you list the material's nominal stress (T_S) and nominal strain in the direction of loading (ε_S) on each line.
- Use the **Pure Volumetric Compression** option to specify volumetric loading test data to include user-defined material compressibility. With **Pure Volumetric Compression**, you define a table field in which you list the material's pressure (p) and the volume ratio, J (current volume/original volume) on each line.

Depending upon the material's type, you can use one or more of these options to define the experimental stress-strain data.

- With the **Arruda-Boyce Test Data**, **Marlow**, and **Foam Test Data** materials, you can only use one of the test data options to define the experimental test data. If you use more than one, when you export or solve your model, the software only writes out the first applicable test data curve. The order in which the software searches for the appropriate test data option depends on the material type, as follows:
  - Arruda-Boyce Test Data: **BIAXIAL Tension** then **UNIAXIAL Tension/Compression**
  - Marlow: **BIAXIAL Tension**, then **PLANAR - Pure Shear**, then **UNIAXIAL Tension/Compression**
  - Foam Test Data: **PLANAR - Pure Shear**, then **UNIAXIAL Tension/Compression**, then **Pure Volumetric Compression**
- With the **Mooney-Rivlin Test Data**, **Neo Hooke Test Data**, **Ogden Test Data**, **Reduced Polynomial**, **Van Der Waals**, and **Yeoh Test Data** material types, you can use up to four of the options to define the experimental test data. If you use more than one, when you export or solve your model, the software writes out the test data option in the following order: **BIAXIAL Tension**, then **PLANAR - Pure Shear**, then **UNIAXIAL Tension/Compression**, then **Pure Volumetric Compression**.

For more information, see:

- **HYPERELASTIC*, **UNIAXIAL TEST DATA*, **BIAXIAL TEST DATA*, **PLANAR TEST DATA*, and **VOLUMETRIC TEST DATA* in the *Abaqus Analysis Keywords Manual*.
- *Hyperelastic behavior of rubberlike materials* in the *Abaqus Analysis User's Manual*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM, idealized part, or part with Abaqus as the specified solver
Toolbar	**Advanced Simulation→Material Properties**

Including data from an external file in Abaqus input files

What is it?

Advanced Simulation now supports the *INCLUDE keyword when you import an Abaqus input file. The *INCLUDE keyword lets you specify the name of an external file that contains a portion of your input file. The external file can include model definition data, comment lines, or even references to other external files. See the *Abaqus Analysis User's Manual* and *Abaqus Keywords Reference Manual* for more information.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

Solver parameter to export group data

What is it?

A new **Write out group data** option has been added to the **Solver Parameters** dialog box. When you export or solve your model, this option controls whether the software exports any groups in your model using the Abaqus *NSET and *ELSET commands.

- If you select this option, the software writes out the group data in your model to the Abaqus input file using the *NSET and *ELSET commands.
- If you clear this option, the software does not write out the group data in your model to the Abaqus input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver
Simulation Navigator	Right-click a solution→**Solver Parameters**

Bolt pre-loads now supported for solid elements

What is it?

In the Abaqus solver environment, you can now define a pre-load on a bolt modeled with solid (continuum) elements. In previous releases, you could only define a bolt pre-load on a bolt modeled with beam (B31) elements.

In the **Bolt Pre-Load** dialog box, use the new **Force on 3D Elements** option in the **Type** list to define a pre-load on solid elements. With this option, you select either elements or faces that define a pre-tension section.

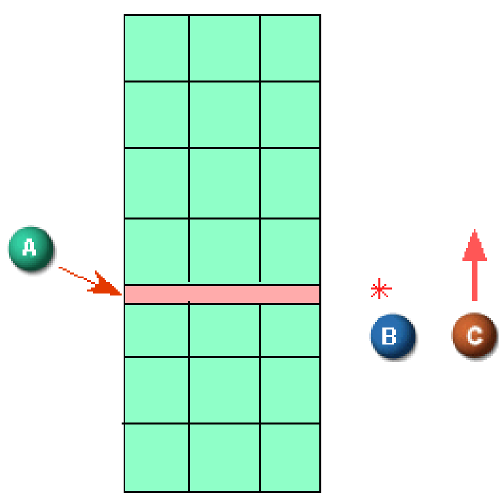
In Abaqus, a pre-tension section is defined as a surface inside the bolt that bisects the bolt. The software transmits the specified pre-load force across the pre-tension section by means of a pre-tension node that you specify. This pre-tension node must not be attached to any element in your model.

> **Note**
>
> If you do not specify a pre-tension node, the software creates a pre-tension node for you when you export or solve your model.

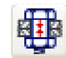
The software applies the load along a vector that is normal to the pre-tension section. The new **Section Normal** option lets you control how the software computes this normal.

- If you select **Average Surface Normal**, the software computes an average normal to the section that faces away from the underlying continuum elements.
- If you select **User Defined**, you can define the vector to specify the normal. This option is useful when the direction in which you want to apply the load is different from the average normal to the pre-tension section.

The following graphic shows an example of a bolt created with solid elements. (A) shows the pre-tension section, (B) shows the pre-tension node, and (C) shows the normal to the pre-tension section.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver
Toolbar	**Advanced Simulation→Bolt Pre-Load** 

Initial clearance values for contact and bolt contact analyses

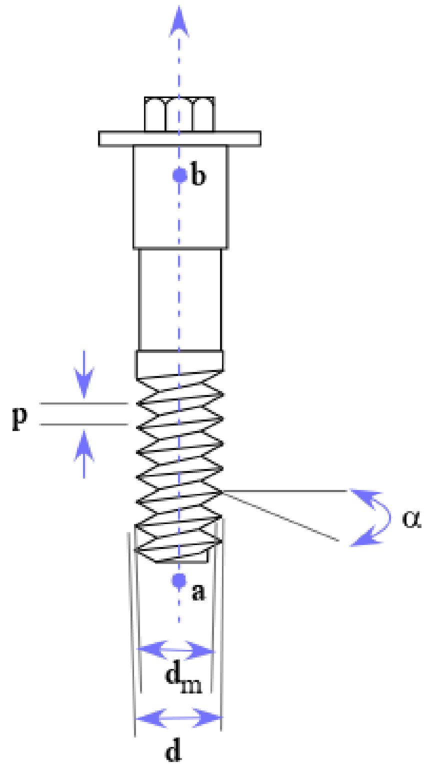
What is it?

You can use the new **Contact with Clearance** and **Bolt Contact with Clearance** simulation objects to define precise initial clearance or overclosure (initial penetration) values for the nodes on the slave (dependent) surface in a contact pair. With both **Contact with Clearance** and **Bolt Contact with Clearance**, the initial clearance or overclosure value you specify overwrites the initial clearance or overclosure value that the software calculates at each slave node.

- Use **Contact with Clearance** to define initial clearances or overclosures when you model contact between two surfaces.
- Use **Bolt Contact with Clearance** to define initial clearances or overclosures when you model contact between a single-threaded bolt and a bolt hole.

Contact with clearance for threaded bolts

The **Bolt Contact with Clearance** dialog box includes additional **Clearance Definition** options that let you model the thread characteristics of a bolt even if detailed thread geometry is not included in the model. These options let you specify details about the bolt threads, including the half thread angle (α), pitch (p, or the thread-to-thread distance), and major (d) and mean (m_d) bolt diameters. You also use the **Bolt Axis** options to define two points (a and b) along the bolt's axis. The software uses these points to generate the bolt's contact normal directions.



Clearance or overclosure value can be uniform or spatially varying

In the **Contact with Clearance** and **Bolt Contact with Clearance** dialog boxes, you can use the **Clearance Definition** options to define the clearance or overclosure value as either uniform or spatially varying for the contact pair. From the **Value** list:

- If you select **Expression**, you can specify a uniform clearance or overclosure value for the contact pair. A positive value indicates a clearance value, and a negative value indicates an overclosure value.
- If you select **Field**, you can specify spatially varying clearances or overclosures. With this option, you use a table field to specify the clearance at a single node or set of nodes on the slave surface. In the table field, the node ID is the independent variable, while the clearance or overclosure value is the dependent variable. You can also specify a **Scale Factor** to apply to the field.

Contact with clearance is supported only in small-sliding contact analyses

You can only use the **Contact with Clearance** or **Bolt Contact with Clearance** commands when you are using the small-sliding contact formulation in your analysis. In Abaqus, you use the *CONTACT PAIR keyword to specify the contact formulation. In Advanced Simulation, you use a **Contact Pair** modeling object to specify the parameters for the *CONTACT PAIR keyword:

1. In the **Contact Pair** dialog box, select **Small** from the **Sliding Type** list to use the small-sliding formulation instead of the finite-sliding formulation.
2. In the **Contact with Clearance** or **Bolt Contact with Clearance** dialog box, use the **Contact Pair** option to associate the **Contact Pair** modeling object with the simulation object.

Associated Abaqus keywords

When you export or solve your model, the software uses the options you specify in the **Contact with Clearance** dialog box to define the *CONTACT PAIR and *CLEARANCE keywords in your Abaqus input file. For more information, see *Adjusting Initial Surface Positions and Specifying Initial Clearances in Abaqus/Standard Contact Pairs* in the *Abaqus Analysis User's Manual* and **CLEARANCE* and **CONTACT PAIR* in the *Abaqus Keywords Reference Manual*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver
Toolbar	**Advanced Simulation→Contact with Clearance** or **Bolt Contact with Clearance**

New thermal conductance boundary condition

What is it?

When you are working with Abaqus as your solver, you can use the new **Surface-to-Surface Thermal Conductance** simulation object to model conductive heat transfer between proximate or contacting surfaces.

In the **Surface-to-Surface Thermal Conductance** dialog box, you can use the **Conductance Dependency** options to model the conductive heat transfer as a function of:

- The clearance between the contacting surfaces (**Clearance** option).
- The contact pressure at the interface between the contacting surfaces (**Pressure** option).
- Both the clearance and the contact pressure (**Clearance and Pressure** option).

In Advanced Simulation, you use fields to define how the heat transfer varies with the clearance and/or contact pressure.

When you export or solve your model, the software uses the options you specify in the **Surface-to-Surface Thermal Conductance** dialog box to define the *GAP CONDUCTANCE keyword in your Abaqus input file. For more information, see *Thermal Contact Properties* in the *Abaqus Analysis User's Manual* and **GAP CONDUCTANCE* in the *Abaqus Keywords Reference Manual*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver and **Thermal** as the selected analysis type
Toolbar	**Advanced Simulation→Surface-to-Surface Thermal Conductance**

Initial support for Abaqus output database files

What is it?

On Windows platforms, this release includes initial support for Abaqus output database files (*.odb) files. In this release, you can:

- Import results into Post-Procressing and create displays.
- Output results from selected analysis steps to an ODB file.

In previous releases, Advanced Simulation only supported the Abaqus results file (*.fil).

Customer default setting necessary to activate ODB support

ODB file support in Advanced Simulation is controlled through a new **ABAQUS File Extension** customer default. To activate ODB support:

1. From the **File** menu, choose **Utilities→Customer Defaults**.
2. In the **Customer Defaults** dialog box, choose **Simulation→ABAQUS**.
3. Click the **Results File** tab and select **ODB** as the **ABAQUS File Extension** default.

   The next time you start NX, Advanced Simulation links the Abaqus libraries that are necessary for working with ODB files.

With the **ABAQUS File Extension** default:

- If you select **FIL**, you can only import and work with Abaqus results (*.fil) files.
- If you select **ODB**, you can import and work with both Abaqus results (.*fil) files and ODB files.

Customer default setting also controls default Abaqus result file type

When you select **ODB** as the **ABAQUS File Extension** default, the software also sets your default output file type for all Abaqus analyses to ODB. With the **ODB** default selected:

- On the **Output** tab in the **Create Solution Step** dialog box, the **Written to ODB** option is selected by default.
- In both the **Results** node in the **Simulation Navigator** and in the **Post-Processing Navigator**, the software looks for files in your current directory that have the .odb extension. If your directory only contains a FIL file and not an ODB file, the software does not display any results.

  Note

  To display results from a FIL file when **ODB** is selected as the **ABAQUS File Extension** default, use the **File→Import→Simulation** command.

Supported ODB version

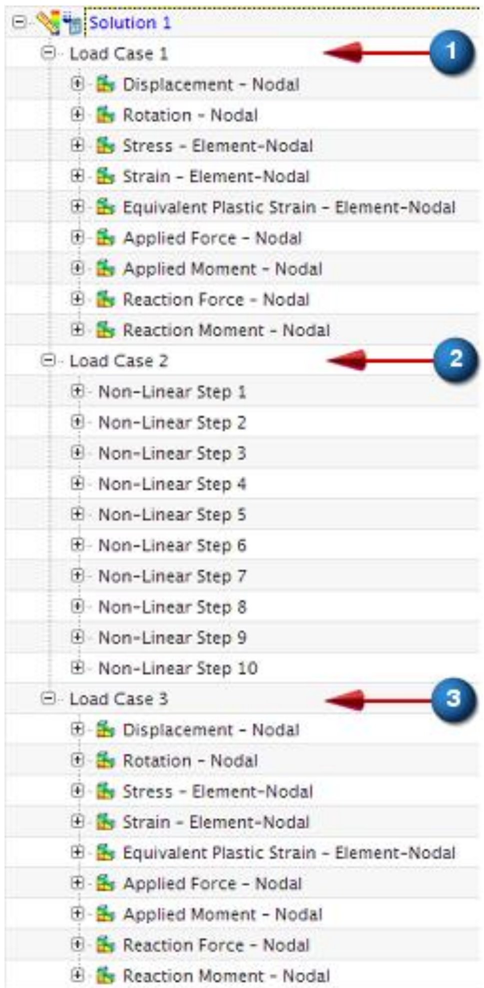
Advanced Simulation currently supports Abaqus 6.8-EF2 version output database (ODB) files on Windows platforms. You can import files in ODB 6.8-EF2 format and earlier. Advanced Simulation exports any results to an ODB file in the 6.8-EF2 format. If you try to import an ODB file from an earlier version of Abaqus, such as Abaqus 6.7-5 or 6.8-2, NX prompts you to convert the file to the 6.8-EF2 version.

Ability to animate across load cases

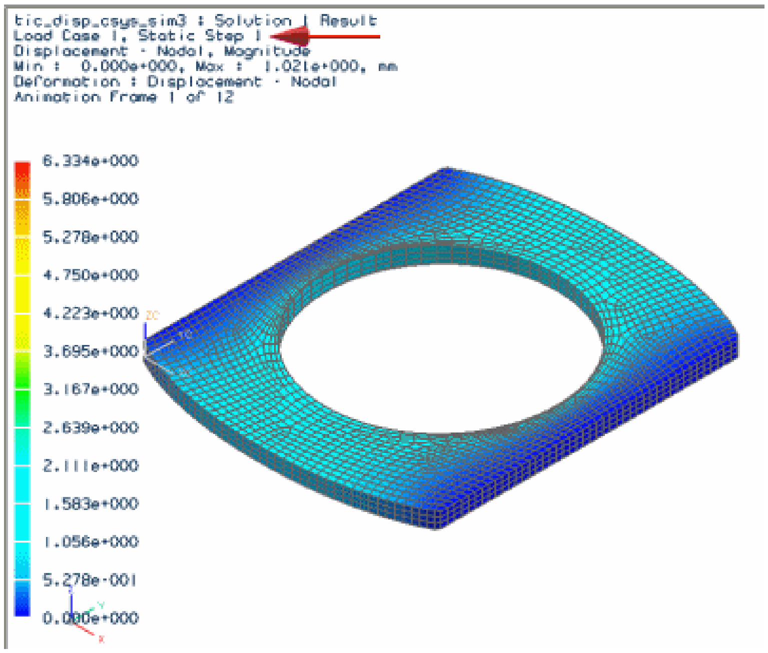
What is it?

For solutions that include multiple load cases, you can now animate the results across the load cases. The **Animation** dialog box includes two options: **Result** and **Iterations**. The **Iterations** option lets you specify a **Start Load Case**, **Start Iteration** (if the starting load case contains iterations), and **End Load Case**, as well as a **Step** increment to determine the number of frames in the animation.

For example, suppose you have three load cases (1, 2, and 3 in the picture below) and you want the animation to start at the first load case and end after the last load case.



In this example, you would choose **Load Case 1** as the **Start Load Case** and **Load Case 3** as the **End Load Case**. The following animated picture shows the result. The first and last load cases each have one static step and the second load case has 10 iterations. The **Step** option is set to **1**, which means the animation includes a frame for each step and iteration between the start and end load case. Therefore, the animation contains 12 frames. Note the load case and iteration number indicated by the arrow in the picture.



Note

In this example, all iterations in Load Case 2 were included automatically because it was not the starting or ending load case. You can specify a **Start Iteration** only if the load case that contains iterations is specified as either the **Start Load Case** or the **End Load Case**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver
Toolbar	**Post-Processing→Animation**

New option to control the import of ELSETs

What is it?

A new option in the **Import Simulation** dialog box lets you control whether the software imports any implicitly defined Abaqus ELSETs in your input file as Advanced Simulation groups. In Abaqus, you can implictly define an ELSET by specifying that all elements of a specific type comprise that set. For example, the following syntax creates an ELSET named set1 that contains all the C3D4 type elements in the model:

```
*ELEMENT, TYPE=C3D4, ELSET=set1,
1,
2
```

In contrast, you create an explicitly defined ELSET by directly listing the elements that form that set. For example, the following syntax creates an ELSET named set2 that contains elements 6, 7, 8, 17, and 26:

```
*ELSET, NAME=set2
6,7,8,17,26
```

If your input file contains implicitly defined ELSETs, you can use the new **Import implict ELSET as group** option in the **Advanced Options** group to control how the software imports those sets.

- If you select **Import implicit ELSET as group**, the software imports any implicitly defined ELSETS in your input file. It uses those ELSETS to create groups.
- If you clear the **Import implicit ELSET as group** check box, the software does not import any implicitly defined ELSETS as groups.

Clearing the **Import implicit ELSET as group** check box can improve the import performance of your input file.

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

ANSYS environment

ANSYS command support enhancements

What is it?

This release includes improved import and export support for ANSYS commands.

Command	Supported Options or Arguments	Unsupported or Partially Supported Options	Location in Advanced Simulation User Interface	For More Information
TB, GASKET	MAT, NPTS, TBOPT	• NTEMP is not supported for loading or unloading	**Materials** dialog box	See **Unsatisfied xref title** and **Unsatisfied xref title**

		curves • EOSOPT is currently unsupported		
TB, HYPER	MAT, NPTS, TBOPT*	• NTEMP and EOSOPT are currently unsupported • The following parameters for TBOPT are supported: MOONEY, OGDEN, BLATZ, FOAM, BOYCE	**Materials** dialog box	See **Unsatisfied xref title**

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation** or **File→Export→Simulation**

New hyperelastic material models

What is it?

This release includes support for a number of new ANSYS hyperelastic material models. Hyperelastic materials let you model materials that are nearly incompressible but which undergo large strains.

Hyperelastic Material Model	Corresponding ANSYS Command
Arruda-Boyce	TB command with the HYPER,,,,BOYCE option
Foam	TB command with the HYPER,,,,FOAM or HYPER,,,,BLATZ options
Gent	TB command with the HYPER,,,,GENT option
Mooney-Rivlin	TB command with the HYPER,,,,MOONEY option
Neo Hooke	TB command with the HYPER,,,,NEO option
Ogden	TB command with the HYPER,,,,OGDEN option
Polynomial	TB command with the HYPER,,,,POLY option

Yeoh	TB command with the HYPER,,,,YEOH option

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM, idealized part, or part with ANSYS as the specified solver
Toolbar	**Advanced Simulation→Material Properties** 

Solver parameter to group face pressure loads

What is it?

A new **Create ANSYS components (CM) for similar loads** option has been added to the **Solver Parameters** dialog box. When you export or solve your model, this option controls whether the software uses the ANSYS CM command to group all elements that have the same face **Pressure** load into a component.

- If you select this option, the software uses the CM command to group all the elements with the same face **Pressure** load and only writes out a single face pressure (SFE) command in your ANSYS input file.
- If you clear this option, the software writes out individual single face pressure (SFE) commands for each element in your ANSYS input file.

You may want to select the **Create ANSYS components (CM) for similar loads** option if you plan to work with your ANSYS input file outside of Advanced Simulation. For example, if you select this option, you can later select the surface pressures by their component name in the ANSYS pre- and post-processing software. Additionally, the **Create ANSYS components (CM) for similar loads** option creates fewer SFE commands, so you may want to select the option to make manually editing your input file easier.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with ANSYS as the specified solver
Simulation Navigator	Right-click a solution→**Solver Parameters**

New options to control the deletion of loads at the end of a step

What is it?

A new **Delete load options** tab has been added to the **Create Solution Step** and **Edit Solution Step** dialog boxes. The options on this tab control whether forces and moments, body loads, and element body loads are kept or deleted by ANSYS at the end of the current solution step. These options correspond to the ANSYS DDELE, FDELE, BFDELE, and BFEDELE commands.

Note

Currently, you can only use these options to delete the loads or constraints on all nodes or elements to which they are applied.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Simulation Navigator	Either right-click a solution and select **Create Step**, or right-click the appropriate solution step and select **Edit Solution Step**

Cumulative loading options now preserved on import

What is it?

When you import an ANSYS input file into Advanced Simulation, the software now preserves the settings of any cumulative option commands (DCUM, FCUM, SFCUM, and BFCUM). In ANSYS, you use these commands to specify whether the software should add (ADD), replace (REPL), or ignore (IGNO) a repeated load or constraint on a particular degree-of-freedom. In previous releases, the software always imported any cumulative option command with a setting of “replace” (REPL), regardless of what you actually specified in the input file. Now, the software correctly preserves the specified cumulative option setting when you import the file.

Note

Advanced Simulation does not import any cumulative options as a **Solution Step** attribute. Instead, the software converts the load or boundary condition data into an equivalent load or boundary condition with the appropriate cumulative option setting.

For example, suppose you have the following constraints on node ID2:

- Step 1: UX = 1mm, UZ= 1.2mm
- Step 2: UX = 1.5mm, UY=1

In step 2 of the analysis, how the software resolves the constraint on node ID2 depends on the setting of the DCUM command:

- If the DCUM option is set to REPL, the resolved constraint for node ID2 is UX = 1.5, UY = 1 (UZ is removed)
- If the DCUM option is set to ADD, the resolved constraint for node ID2 is UX = 2.5, UY = 1, UZ = 1.2

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Menu	**File→Import→Simulation**

Support for importing loads and constraints from binary files

What is it?

When you import an ANSYS binary file into Advanced Simulation, the software now imports any loads (forces/moments) and constraints (displacements and temperatures) defined in that file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Menu	**File→Import→Simulation**

LS-DYNA environment

LS-DYNA keyword support

What is it?

Advanced Simulation now provides export support for the following LS-DYNA keywords:

Keyword	For More Information
*CONTROL_SHELL	See **Unsatisfied xref title**
*INTEGRATION_SHELL	See **Unsatisfied xref title**
*MAT_COMPOSITE_DAMAGE	See **Unsatisfied xref title**
*PART_COMPOSITE	See **Unsatisfied xref title**
*SET_NODE_LIST	See **Unsatisfied xref title**
*SET_NODE_BEAM	See **Unsatisfied xref title**
*SET_DISCRETE	See **Unsatisfied xref title**
*SET_SHELL_LIST	See **Unsatisfied xref title**
*SET_SOLID	See **Unsatisfied xref title**
*SET_TSHELL	See **Unsatisfied xref title**

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with LS-DYNA as the specified solver
Toolbar	**File→Export→Simulation**

Support for laminates

What is it?

You can now use the NX Composite Laminates capabilities to create laminates in an LS-DYNA model.

Laminate physical properties enhancements

For LS-DYNA models, two new laminate options have been added to the **Type** list in the **Physical Property Tables Manager** dialog box:

- The **PART (laminate shell)** type for shell elements (*ELEMENT_SHELL).
- The **PART (laminate thick shell)** type for solid elements (*ELEMENT_TSHELL).

Select the appropriate option from the **Type** menu and then click **Create** to open the **Laminate Modeler** dialog box.

In the **Laminate Modeler** dialog box, you can then use the new **Keyword Option** menu to select either the ***PART_COMPOSITE** or ***INTEGRATION_SHELL** keywords to define the laminate properties.

- If you select ***PART_COMPOSITE**, you do not need to create a **SECTION_SHELL** type of modeling object. When you export or solve your model, the software writes the ply data (thicknesses, material IDs, and angle) you define in the **Laminate Modeler** dialog box out as a *PART_COMPOSITE keyword.
- If you select ***INTEGRATION_SHELL**, you must create a **SECTION_SHELL** type of modeling object. When you export or solve your model, the software converts ply thicknesses that you define in the **Laminate Modeler** dialog box to weighting factors and writes out an *INTEGRATION_SHELL keyword. The number of plies is the number of integration points through the thickness. When you export or solve your model, the software writes the ply orientation angle to the *SECTION_SHELL keyword. For materials:
  - If you select the **Use NX Composite Material** option, the software does not write out any material information when you export or solve your model. This is because LS-DYNA composite materials are not currently supported in Advanced Simulation.
  - You can use the **LSDYNA Composite Material ID** box to specify the ID of an LS-DYNA composite material that exists outside of NX.

For more information, see the *LS-DYNA Keyword User's Manual*.

Inheriting a laminate from a layup

Currently, when you inherit a laminate from a layup, only the *PART_COMPOSITE option is supported.

Inheriting material orientation from a layup

For shell (2D) elements, you can specify that they inherit their material orientation from the layup. To do this:

- In the **Mesh Associated Data** dialog box, select **Set Material Orientation** from the **Keyword Option** list.
- From the **Material Orientation Definition** list, select **MCID Inherited from Layup**.
- Click **OK**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with LS-DYNA as the specified solver

Export support for groups

What is it?

If you create groups in your FEM file, you can now export those groups when you export your model to an LS-DYNA keyword file. To facilitate this, the software now exports the following LS-DYNA keywords:

- *SET_NODE_LIST
- *SET_NODE_BEAM
- *SET_DISCRETE
- *SET_SHELL_LIST
- *SET_SOLID
- *SET_TSHELL

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with LS-DYNA as the specified solver
Toolbar	**File→Export→Simulation**

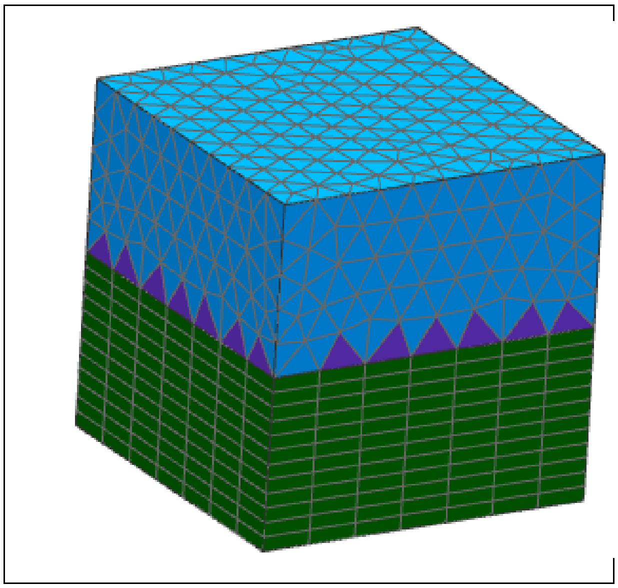
Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal

General capabilities

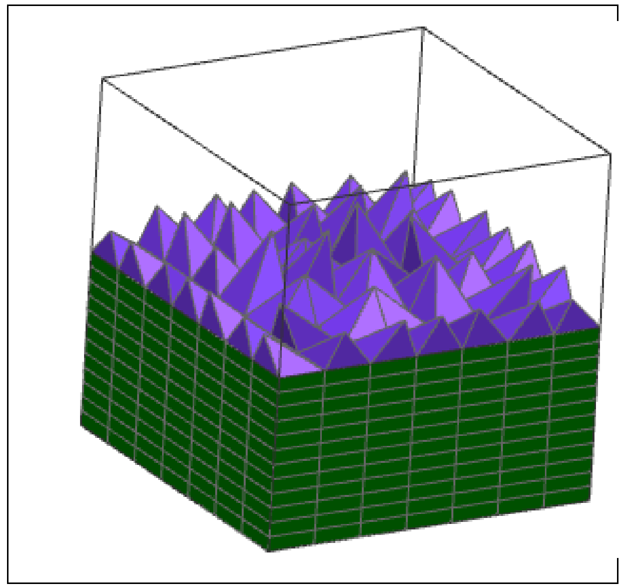
3D transitional elements

What is it?

When you have both a hexahedral mesh and a tetrahedral mesh, you can specify the use of pyramid elements as transitional elements from one mesh to the other as shown in the following figures.



Visible tetrahedral mesh



Invisible tetrahedral mesh

The purple pyramid elements are the transitional elements between the hexahedral mesh in green and the tetrahedral mesh in blue.

To specify the use of pyramid elements, you must select the **Transition with Pyramid Elements** option in the **3D Tetrahedral Mesh** dialog box. This option is available only when you have a hexahedral mesh present next to the solid body which will be meshed with tetrahedral elements and a mesh mating condition between the two bodies.

Why should I use it?

Use the pyramid elements for the transition between hexahedral and tetrahedral meshes to improve flow results for computational fluid dynamics simulations.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling

		Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal
		Advanced Thermal
Flow	Flow	
	Advanced Flow	
Coupled Thermal-Flow	Thermal-Flow	
	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar →**3D Tetrahedral Mesh** 
Menu	**Insert** → **Mesh** → **3D Tetrahedral Mesh**
Location in dialog box	**Mesh Parameters** group → **Transition with Pyramid Elements**

Material orientation vector and element thickness for individual elements

What is it?

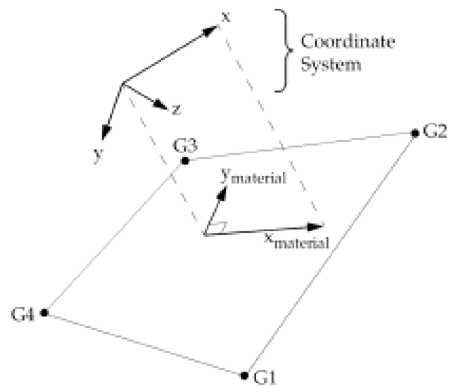
You can use the **Element Associated Data** command to define a material orientation vector and element thickness for individual elements with the following physical properties:

- **Thin Shell**
- **Multi-layer Uniform Shell** with orthotropic material

This allows you to define material orientation and element thickness that vary across different elements. In previous releases, you could only specify a uniform set of properties across an entire mesh.

You can define material orientation property by specifying one of the following:

- The **Coordinate System** — The software uses the X-axis of the selected coordinate system as the material orientation vector. The X-axis vector is projected onto the plane of the selected element to determine the material orientation.
- The **Vector Projection** — The software projects the specified vector onto the plane of the selected element to determine the material orientation (the vector becomes the X-axis).



You must set the corner node thickness to specify the thickness of the element. This shell thickness overrides the thickness defined in the mesh.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling
		Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal
		Advanced Thermal
Flow	Flow	
	Advanced Flow	
Coupled Thermal-Flow	Thermal-Flow	
	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains elements for which element associated data is supported.
Toolbar	**Element Operations → Element Associated Data**
Menu	**Edit →Element →Modify Associated Data**

Report thermal results on both sides of 2D Shell elements

What is it?

You can now specify on which side of 2D shell elements you want to calculate thermal results when using a **Report** simulation object of following types:

- **Per Element**
- **Per Region**
- **Between Regions**
- **Heat Maps**

You can ask for results on the following sides of 2D shell elements:

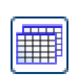
- **Top and Bottom**
- **Top**
- **Bottom**

For a **Between Regions** type of **Report**, you can separately set the side for elements in the primary and secondary regions.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
Flow	Flow Advanced Flow	
Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container → **New Simulation Object** → **Report**
Toolbar	**Advanced Simulation** toolbar → **Report** 

Group import

What is it?

When you import **NX THERMAL / FLOW**, **NX SPACE SYSTEMS THERMAL**, **NX ELECTRONIC SYSTEM COOLING**, or **IDEAS UNV** simulations, the software now also imports any groups containing elements and nodes.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
Flow	Flow Advanced Flow	
Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or SIM with **NX THERMAL / FLOW**, **NX SPACE SYSTEMS THERMAL**, or **NX ELECTRONIC SYSTEM COOLING** solver
Menu	**File → Import → Simulation**

Thermal Coupling simulation objects name change

What is it?

Some simulation objects were renamed:

Old name	New name
Advanced Thermal Coupling	**Thermal Coupling — Advanced**
Convection Coupling	**Thermal Coupling — Convection**
Radiation Thermal Coupling	**Thermal Coupling — Radiation**

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling (**Thermal Coupling — Radiation** only) Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal (**Thermal Coupling — Radiation** only) Advanced Thermal
Coupled Thermal-Flow	Thermal-Flow (**Thermal Coupling — Radiation** only) Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container and choose **New Simulation Object**
Toolbar	**Advanced Simulation** toolbar → **Simulation Object Type** 

Dialog box text changes

What is it?

The following minor text changes were made in this release:

Dialog box name	Group name	Old text	New text
Flow Boundary Condition	**Flow Direction** and **Flow Return**	**Normal to Tangential Velocity Angle**	**Normal to Resultant Velocity Angle**
Particle Injection	**Parameters**	**Mass per Seconds**	**Mass per Unit Time**
Thermo-Optical Properties — Advanced	**Infrared Properties**, **Solar Properties**, and **Non Gray Properties**	**Transparency**	**Transmissivity**

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling
		Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal
		Advanced Thermal
Flow	Flow	
	Advanced Flow	
Coupled Thermal-Flow	Thermal-Flow	
	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar → **Simulation Object Type**  **Advanced Simulation** toolbar → **Modeling Objects** 

Orthotropic thermal conductivity in cylindrical or spherical coordinates

What is it?

You can now specify orthotropic materials with thermal conductivity defined in cylindrical or spherical coordinates. In the 2D and 3D **Mesh Collector** dialog boxes, when you specify an orthotropic material, you have the following choices for the **Material Orientation Type**:

- **Cartesian** to specify that the orthotropic conductivity is defined in Cartesian coordinates.
- **Cylindrical** to specify that the orthotropic conductivity is defined in cylindrical coordinates.
- **Spherical** to specify that the orthotropic conductivity is are defined in spherical coordinates.

The following table shows the correspondence between the **Thermal Conductivity X**, **Thermal Conductivity Y**, and **Thermal Conductivity Z** values you enter in the **Materials** dialog box and their values in all three coordinate systems:

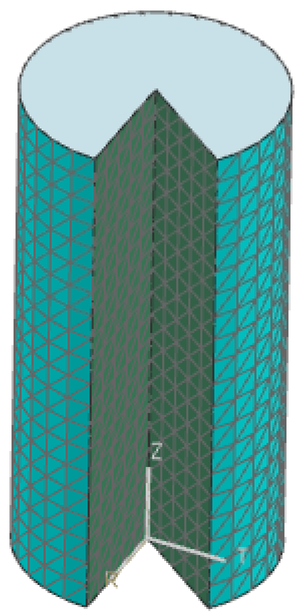
Material Orientation Type	Thermal Conductivity X	Thermal Conductivity Y	Thermal Conductivity Z
Cartesian	k_x	k_y	k_z

Cylindrical	k_r	k_θ	k_z
Spherical	k_r	k_θ	k_φ

For 2D mesh collectors, when you select **Cylindrical** or **Spherical** as the material orientation type, you must also specify the **Main Conduction Plane and Direction** for the heat transfer. The software determines the three components of the thermal conductivity as follows:

- The software projects on the shell element the main direction vector, and the conductivity in that direction is the main direction vector's conductivity.
- The conductivity in the direction of the element normal is the main conduction plane conductivity.
- The third component of the conductivity is the remaining conductivity and its direction is the remaining direction to complete the right-hand side coordinate system.

For example, suppose you want to model temperature distribution on the following cylindrical object with orthotropic thermal conductivity using thin shell elements. You drape the cylinder with a material that has the thermal conductivity of 0.1 W/m°C in the radial direction on the dark green surfaces and also in the angular direction on the cyan surface (**Thermal Conductivity X**), that has the thermal conductivity of 10 W/m°C in the axial direction (**Thermal Conductivity Y**) and 0.0001 W/m°C in the remaining



direction (**Thermal Conductivity Z**).

You need to create two **2D Mesh Collectors** with the **Cylindrical** option for the **Material Orientation Type** list and:

- For the mesh collector with cyan color, select the **R plane, Theta direction** option for the **Main Conduction Plane and Direction** list.
- For the mesh collector with dark green color, select the **Theta plane, R direction** option for the **Main Conduction Plane and Direction** list.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling
		Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal
		Advanced Thermal
Flow	Flow	
	Advanced Flow	
Coupled Thermal-Flow	Thermal-Flow	
	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM
Toolbar	**Advanced Simulation → Mesh Collector**
Menu	**Insert → Mesh Collector**
Location in dialog box	**Material** group

UNV import of axisymmetric thermal models

What is it?

You can now import UNV models that contain axisymmetric elements into a thermal model.

Previously UNV models containing axisymmetric elements were always imported into an axisymmetric thermal model.

When importing UNV models, if axisymmetric elements are detected, the following message appears in the **Model Information** text box of the **Import Options** dialog box:

```
Axisymmetric elements were detected
```

The **Set Simulation Analysis Type To** list has the following options:

- **Coupled Thermal-Flow** (appears for Thermal and Flow or Electronic Systems Cooling)
- **Thermal** (appears for Space Systems Thermal)
- **Axisymmetric Thermal**

If you select **Coupled Thermal-Flow** or **Thermal**, the software imports axisymmetric elements as normal elements and creates:

- A generic entity named **Converted_Axi_Shells** to reference axisymmetric shell elements.
- A generic entity named **Converted_Axi_Solids** to reference axisymmetric solid elements.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling
		Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Coupled Thermal-Flow	Thermal-Flow
		Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar →**Modeling Objects**  →**Advanced Parameters** → **Create**
Menu	**Insert** → **Modeling Objects** → **Advanced Parameters** → **Create**

Thermal capabilities

Articulation modeling enhancement

What is it?

You now have more flexibility when defining time step options for the articulation movement. You can select one of the following four options in the **Time Step Option** list of the **Articulation Parameters** group on the Transient Setup page of the **Create Solution** or **Edit Solution** dialog box:

- **At Constant Time Intervals** — you control the starting and ending times for the articulation as well as the calculation interval.
- **Total Number** — you control the starting and ending times for the articulation as well as the number of time steps.
- **At Specified Times** — you specify the articulation movement times in a list of the **Articulation Times** dialog box by clicking  .
- **Interval versus Time Table** — you specify a table containing the desired calculation intervals at specified times. For example:

Time	Interval
0.0	1.0
10.0	2.0
20.0	3.4

The thermal calculation for the articulation movement is performed at following times:

0.0	1.0	2.0	3.0	4.0	5.0	6.0	7.0	8.0	9.0	10.0	12.0	14.0	16.0	18.0	20.0

The last row of the table indicates the articulation end time; the calculation interval is a dummy number.

In previous releases, the only available time step control option was **At Constant Time Intervals**.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
Coupled Thermal-Flow	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the solution node and choose **Solve**
Toolbar	**Advanced Simulation** toolbar → **Solve**  → **Edit Solution Attributes**

Hemicube method enhancements

What is it?

Two enhancements have been made to the algorithm for the view factor calculation using the Hemicube method. The algorithm now includes

- A more robust algorithm that determines the near and far cutting planes for perspective projections in rendering-based view factor calculations.
- A filtering algorithm that addresses situations where numerical noise could create spurious view factors, which, even though very small, could cause enclosure leakages.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow	

Element subdivision in Hemicube view factor calculation

What is it?

When you use the Hemicube method to calculate view factors, you can now unconditionally enforce the use of elemental subdivisions.

By default, the element subdivision is performed when one of the view factors seen through:

- The top viewing plane is greater than 0.03.
- The side viewing plane is greater than 0.3.

To ensure that the software always performs the element subdivision, you can set the new **ENFORCE MESH IN HEMIVIEW** advanced parameter.

Note

You set the **Element Subdivision** parameter in one or more of these places:

- The Radiation Parameters page of the **Solver Parameters** dialog box
- The **Radiation** simulation object
- The **Radiative Element Subdivision** simulation object

Why should I use it?

Use the new parameter if there are any significant deviations of view factor sums from unity. This option ensures higher accuracy of view factor results at the expense of increased computation time.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal Advanced Thermal
Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** → **Modeling Objects**  → **Advanced Parameters** → **Create**
Menu	**Insert** → **Modeling Objects** → **Advanced Parameters** → **Create**

Enhancements to Thermal Coupling — Convection simulation object

What is it?

In this version, there are two new enhancements to the **Thermal Coupling — Convection** simulation object:

- The **Coupling Resolution** option to control the accuracy of the coupling is now available for all three types:
  - **Forced Convection Coupling**
  - **Free Convection Coupling**
  - **Across Gap Convection Coupling**

  Two different approaches are available: **One to One** resolution and a series of five resolution settings ranging from **Coarse** to **Finest**. The **Coupling Resolution** option gives you greater control for the convective thermal coupling calculation.
- For the **Free Convection Coupling** type, the **Secondary Region** is renamed **Fluid Ducts**. The convecting fluid is the fluid in the 1D duct network.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
Flow	Advanced Flow	
Coupled Thermal-Flow	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container → **New Simulation Object** → **Thermal Coupling — Convection**
Toolbar	**Advanced Simulation** toolbar → **Thermal Coupling — Convection** 

Enhancement to Radiative Element Subdivision simulation object

What is it?

When selecting elements to which to apply the local **Radiative Element Subdivision** settings, the **Element Selection Filtering** is enhanced. You can now select from the **Filter Type** list:

- **1D Elements** (previously available) — lets you select only 1D elements.
- **2D Elements** (previously available) — lets you select only 2D elements.
- **Element Edges** (new) — lets you select 1D elements and the edges of 2D elements.
- **Element Faces** (new) — lets you select 2D elements and the faces of 3D elements.

You must first set **Type Filter** on the Selection bar to **Element**.

Why should I use it?

Use these new options to easily select elements to which you want to apply local element subdivision for radiation calculations.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
Coupled Thermal-Flow	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container → **New Simulation Object** → **Radiative Element Subdivision**
Toolbar	**Advanced Simulation** toolbar → **Radiative Element Subdivision** 

Coupling Resolution option

What is it?

You can control the accuracy of the coupling using the **Coupling Resolution** option.

The **Coupling Resolution** option is added to the following simulation objects:

- **Electrical Coupling** type of **Joule Heating**
- **Surface to Surface Contact**
- **Thermal Coupling — Advanced**

Two different approaches are available: **One to One** resolution, and a series of five resolution settings ranging from **Coarse** to **Finest**.

- With **One to One** resolution, the solver calculates a single conductance from each primary element to the nearest secondary element. Some inaccuracy may be introduced in the exact location of the conductance on the larger surface, but the general trend of in-plane conductance is maintained. This is adequate for most applications.

  With **One to One** resolution, you can improve the coupling's accuracy by:

  – Using a finer mesh on the secondary surface than on the primary surface.

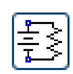
  – Matching the meshes on the two surfaces so that there is a one to one correspondence between nodes, and so that each primary element shadows only one secondary element. The nodes do not have to match perfectly.

- With the settings **Coarse** through **Finest**, each primary element is divided into a progressively larger number of sub-elements. For each sub-element, an area proportional conductance is created to the nearest of the secondary elements. Once conductances from each of the sub-elements are calculated, they are merged and parallel conductances are combined. The result is area-proportional couplings which are distributed among the secondary elements based on element overlap.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling (**Joule Heating** and **Surface to Surface Contact** only) Advanced Thermal/Flow with ESC
NX Thermal and Flow	Thermal	Thermal (**Surface to Surface Contact** only) Advanced Thermal
Coupled Thermal-Flow	Thermal-Flow (**Surface to Surface Contact** only) Advanced Thermal-Flow	
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container → **New Simulation Object**→ **Joule Heating**, **Surface to Surface Contact**, or **Thermal Coupling — Advanced**
Toolbar	**Advanced Simulation** toolbar → **Joule Heating**  , **Surface to Surface Contact**  , or **Thermal Coupling — Advanced** 
Location in dialog box	**Additional Parameters** group

Only Connect Overlapping Elements option

What is it?

You can establish couplings only along a path that is normal to each primary sub-element using the **Only Connect Overlapping Elements** option.

The **Only Connect Overlapping Elements** option is added to the following simulation objects:

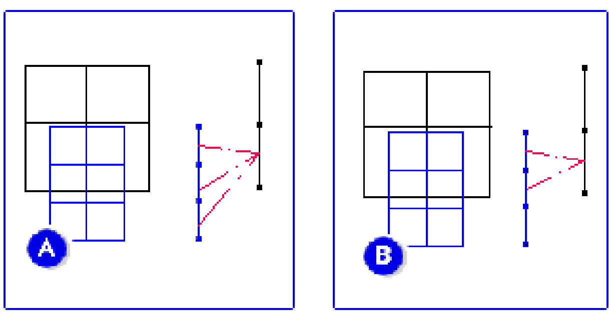
- **Electrical Coupling** type of **Joule Heating**
- **Surface to Surface Contact**

Why should I use it?

The **Only Connect Overlapping Elements** option can be useful when some primary elements have no overlapping secondary elements.

- When this option is not selected, the software establishes a conductance from each primary sub-element to the nearest secondary element as shown in (A) below.
- When you select this option, the software only establishes couplings along a path that is normal to each primary sub-element. No conductance is created for primary sub-elements that have no overlapping secondary element, as shown in (B) below.

Primary elements are shown in blue.



A	**Only Connect Overlapping Elements** cleared.
B	**Only Connect Overlapping Elements** selected.

Note

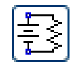
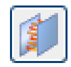
Secondary elements with no overlapping primary elements are never connected.

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	Thermal	Thermal (**Surface to Surface Contact** only) Advanced Thermal
Coupled Thermal-Flow	Thermal-Flow (**Surface to	

	Surface Contact only) Advanced Thermal-Flow	
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container → **New Simulation Object** → **Joule Heating** or **Surface to Surface Contact**
Toolbar	**Advanced Simulation** toolbar → **Joule Heating**  or **Surface to Surface Contact** 
Location in dialog box	**Additional Parameters** group

Duct modeling capabilities

Duct Opening enhancement

What is it?

When you specify external conditions for a **Duct Opening** type of the **Duct Flow Boundary Conditions** simulation object, you can now specify only the pressure of the fluid or you can also specify the temperature of the fluid. In previous versions, you needed to specify both the pressure and the temperature at the same time.

A **Temperature** check box now appears when you select **Specify** from the **External Conditions** list.

- By selecting the **Temperature** check box, you specify the **Temperature Value** at the duct opening.
- By clearing the **Temperature** check box, the temperature of the fluid at the duct opening is the **Fluid Temperature** value defined in the Ambient Conditions page of the **Create Solution** or the **Edit Solution** dialog box.

In both cases, you must specify the **Total Pressure — Gauge** value for the pressure of the fluid at the duct opening.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal

Flow	Advanced Flow
Coupled Thermal-Flow	Advanced Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container → **New Simulation Object** → **Duct Flow Boundary Conditions**
Toolbar	**Advanced Simulation** toolbar → **Duct Flow Boundary Conditions**

New results on duct elements

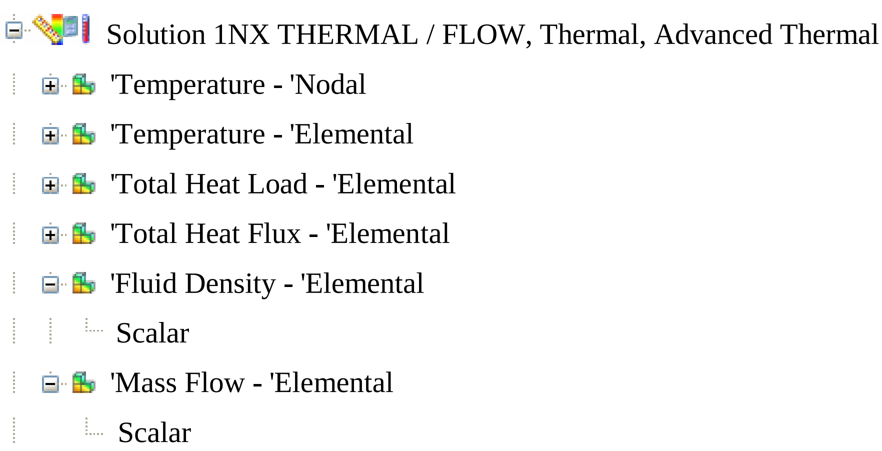
What is it?

On duct elements, in addition to the existing results, you can now recover:

- Density
- Mass flow

To recover these results from the analysis, select **Densities** or **Mass Flows** in the **Duct Flow** group of the Results Options page of the **Create Solution** or **Edit Solution** dialog boxes.

After solving, in the **Post Processing Navigator**, a **Fluid Density — Elemental** and/or **Mass Flow — Elemental** result type appears under the Solution results node as seen in the following graphic.



Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC

NX Space Systems Thermal	Thermal	Space Systems Thermal
NX Thermal and Flow	Thermal	Advanced Thermal
Flow	Advanced Flow	
Coupled Thermal-Flow	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the solution node and choose **Edit Solution**
Toolbar	**Advanced Simulation** toolbar → **Solve** → **Edit Solution Attributes**
Menu	**Analysis** → **Solve** → **Edit Solver Attributes**
Location in dialog box	**Duct Flow** group on the Results Options page

Flow capabilities

Export flow model and boundary conditions to CGNS

What is it?

You can export nodes, elements, and the following boundary condition data in the CFD General Notation System (CGNS) format:

- Temperature
- Inlet information: velocity, mass flow, volume flow, or pressure rise
- Outlet information: velocity, mass flow, volume flow, or pressure rise
- Opening information: pressure

For the element on free faces where you apply boundary conditions, the software creates boundary elements with a normal pointing into the fluid. Any pyramid elements are converted to wedge elements.

CGNS export does not support table data. For temperature, velocity, mass flow, volume flow, or pressure varying in time, the software exports the boundary condition into the CGNS format with a value equal to 0. The software does not export any boundary condition values that are defined with NX spatial fields.

CGNS is a standard format for recording computational fluid dynamics (CFD) analysis data.

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	Advanced Flow
	Coupled 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**

Relaxation factor for turbulence equations

What is it?

You can now specify the **Turbulence** relaxation factor in the **Relaxation Factors** group on the 3D Flow Solver page of the **Solver Parameters** dialog box.

The solver applies this relaxation parameter to the turbulence equations when you set one of the following two-equation models in the **Turbulence Models** option on the Solution Details page of the **Create Solution** or **Edit Solution** dialog boxes:

- **K-Epsilon**
- **Shear Stress Transport-SST**
- **K-Omega**

The value that you set for the turbulence relaxation factor should always be positive and less than or equal to 1.

Reducing this value slows down the convergence rate and should be used when convergence cannot be achieved otherwise.

Why should I use it?

This feature is particularly useful when divergence occurs early in the iterative process soon after the turbulence equations become active. The main purpose of this relaxation factor is to prevent wrong values of k or ε or ω in or near the boundary layer from affecting the other variable.

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
Simulation Navigator	Right-click the Solution node → **Solver Parameters**
Toolbar	**Advanced Simulation** → **Solve**  → **Edit Solver Parameters**
Menu	**Analysis** → **Solve** → **Edit Solver Parameters**
Location in dialog box	**Relaxation Factors** group on the 3D Flow Solver page

Spectral attenuation report

What is it?

When you solve a flow or coupled transient analysis, you can now request the creation of spectral attenuation graphs.

The spectral attenuation, S_A, is a function of frequency, f, and is given by this equation:

$$S_A(f) = 10log_{10}\left(\frac{|P_{em}(f)P_{re}^*(f)|}{|P_{re}(f)P_{re}^*(f)| + 10^{-10}}\right)$$

Where:

$P_{em}(f)$ is the complex number of the fast Fourier transform of the emitted pressure at frequency f.

$P_{re}(f)$ is the complex number of the fast Fourier transform of the received pressure at frequency f.

$P^*_{re}(f)$ is the complex conjugate of $P_{re}(f)$.

In the **Between Regions** type of **Report** simulation object, select:

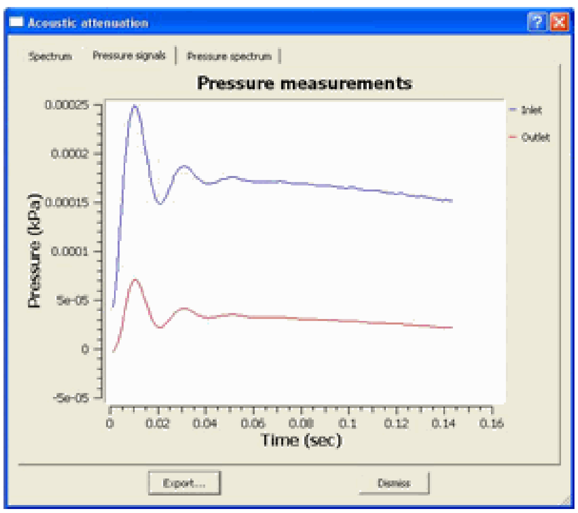
- The region that has the emitted signal as your **Primary Region**.
- The region that has the received signal as your **Secondary Region**.
- The **Spectral Attenuation** check box.

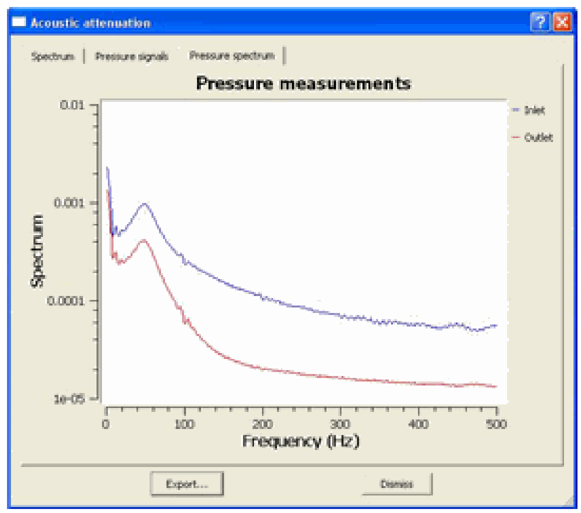
Note

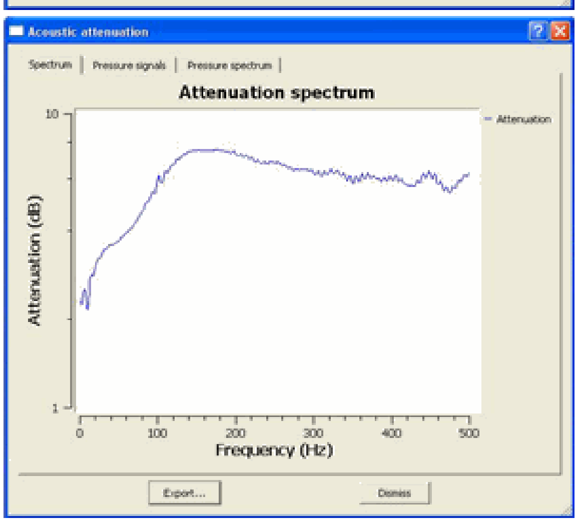
To create a spectral attenuation report, you must select constant time steps, because the software calculates the discrete frequencies using the time step value it assumes to be constant. To select constant time steps, select **Constant** from the **Time Step Option** list on the Transient Setup page of the **Create Solution** or **Edit Solution** dialog boxes to let the software calculate the discrete frequencies.

After the solve, the software saves the following graphs in the run directory:

- The emitted and received pressures as functions of time in *flowsppsig.png*.
- The fast Fourier transform of the emitted and received pressures as functions of frequency in *flowsppspec.png*.
- The spectral attenuation as a function of frequency in *flowspatt.png*.







The above data is also saved into the *GroupReport.htm* and *GroupReport.csv* files.

Why should I use it?

You should request the creation of the spectral attenuation report when you want to simulate noise reduction or acoustic wave propagation.

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
Simulation Navigator	Right-click the Simulation Object container → **New Simulation Object** → **Report**
Toolbar	**Advanced Simulation** → **Report**
Location in dialog box	**3D Flow Data** group

Periodic fluid domain mesh

What is it?

You can now create periodic fluid domain meshes when applying periodic boundary conditions.

To create a periodic fluid domain mesh, you need to:

1. Create a 3D periodic model.
2. Create a **Fluid Domain** simulation object.
3. Create a **Periodic Boundary Condition** simulation object.
4. Create *meshMatching.dat* file in the same folder as the Simulation file with the following format:

```
################################################################################
############
# This file should be used to define periodic boundary conditions to the
mesher.
```

```
# The format is as follow:
#
# Periodicity LBC NAME
# Trans_x Trans_y Trans_z Axis_x Axis_y Axis_z Center_x Center_y Center_z Theta
#
# Where:
#
# Trans Translation
# Axis Axis of rotation
# Center Center of rotation
# theta Angle of rotation (degrees)
#
###############################################################################
############
Periodic Boundary Condition(1)
0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
Periodic Boundary Condition(2)
0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
Periodic Boundary Condition(3)
0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
```

The software generates the periodic fluid domain mesh when you solve the model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the Simulation Object container → **New Simulation Object** → **Fluid Domain**
Toolbar	**Advanced Simulation** → **Fluid Domain** 

Bivariate fluid properties

What is it?

You can have the following fluid properties varying with both temperature and pressure:

- Density
- Specific heats at constant pressure
- Specific heats at constant volume
- Dynamic viscosity
- Conductivity
- Thermal expansion coefficient

Use a **Generic Entity** modeling object to specify bivariate fluid properties. For complete instruction on how to specify bivariate fluid properties, see Define a bivariate fluid material property.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar → **Modeling Object**  → **Generic Entity** → **Create**
Menu	**Insert** → **Modeling Objects** → **Generic Entity** → **Create**

Convection Properties option

What is it?

The behavior of the **Convection Properties** option has changed. You can now define convection properties even when you select the **Slip Wall** check box in the following dialog boxes:

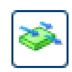
- **Flow Surface**
- **Flow Blockage**
- **Create / Edit Solution**

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling Advanced Thermal/Flow with ESC
NX Thermal and Flow	Flow	Flow Advanced Flow
Coupled Thermal-Flow	Thermal-Flow Advanced Thermal-Flow	

Where do I find it?

Simulation objects

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container →**New Simulation Object** → **Flow Surface** / **Flow Blockage**
Toolbar	**Advanced Simulation** toolbar → **Flow Surface**  / **Flow Blockage** 
Location in dialog box	**Flow Surface** dialog box → **Convection Properties** **Flow Blockage** dialog box → **Solid Blockage** type → **Convection Properties**

Create or edit simulation

Application	Advanced Simulation
Simulation Navigator	• Right-click the simulation → **New Solution** • Right-click the solution → **Edit Solution**
Location in dialog box	3D Flow page → **Default Convection Properties**

What is it?

You can now model gases using the Redlich-Kwong Real Gas Equation of State:

$$P = \frac{RT}{V_m - b} - \frac{a}{\sqrt{T}V_m(V_m + b)}$$

where

- P is the pressure of the gas.
- T is the temperature of the gas.
- V_m is the molar volume of the gas.
- R is the universal gas constant (8.314472 J/(mol·K)).

The constants a and b are defined as:

$$a = 0.42748\frac{R^2 T_C^{2.5}}{P_C}$$

$$b = 0.08662\frac{RT_C}{P_C}$$

where

- P_C is the critical pressure of the gas.
- T_C is the critical temperature of the gas.

You use an **Advanced Parameters** modeling object to specify the Redlich-Kwong real gas equation of state. For complete instructions on how to specify the Redlich-Kwong equation, see Define a gas using the Redlich-Kwong equation of state.

Why should I use it?

Use the Redlich-Kwong Real Gas Equation of State at high pressure as it is more realistic than the ideal gas law.

The Redlich-Kwong equation should be used when the ratio of the pressure to the critical pressure is less than about one-half of the ratio of the temperature to the critical temperature:

$$\frac{P}{P_C} < \frac{T}{2T_C}$$

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	Advanced Flow
Coupled Thermal-Flow	Advanced Thermal-Flow	

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar →**Modeling Objects**  →**Advanced Parameters** → **Create**
Menu	**Insert** → **Modeling Objects** → **Advanced Parameters** → **Create**

Mapping capabilities

Automatic detection of mapping source analysis type

What is it?

In the **Create Solution** dialog box with the **Solver** list set to **NX THERMAL / FLOW** or **NX ELECTRONIC SYSTEMS COOLING**, when you select the **Mapping** from the **Analysis Type** list, the source model analysis type is detected automatically from the *BUN* file associated with the source analysis.

In the Mapping Details page of the **Create Solution** or **Edit Solution** dialog boxes, the **Source Model Analysis Type** field is now non-editable.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Mapping	Thermal-Flow
NX Thermal and Flow	Mapping	Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the solution node and choose **Edit Solution**
Toolbar	**Advanced Simulation** toolbar → **Solve**  → **Edit Solution Attributes**
Menu	**Analysis** → **Solve** → **Edit Solver Attributes**
Location in dialog box	**Data Source and Destination** group on the Mapping Details page

What is it?

All **Mapping** and **Axisymmetric Mapping** solutions in **NX THERMAL / FLOW**, **NX SPACE SYSTEMS THERMAL**, and **NX ELECTRONIC SYSTEMS COOLING** solvers allow you to automatically create a complete NX solution with all the appropriate steps, subcases, and loads defined for the Nastran, ANSYS, and Abaqus solvers.

To create complete NX solutions, in the Optional Output page of the **Create Solution** or **Edit Solution** dialog box, select the **Create Nastran Solution**, **Create Ansys Solution**, or **Create Abaqus Solution** check box.

The mapping solver creates the following NX solutions:

- **SESTATIC 101 — Single Constraint** for NX Nastran only
- **Linear Statics** for ANSYS
- **General Analysis** for Abaqus

When the **Analysis Type** is **Mapping**, the loads the mapping solver creates depends upon the **Source Model Analysis Type** and the setting of the **Data to Map** option (Mapping Details page of the **Create Solution** and **Edit Solution** dialog boxes).

- When the **Source Model Analysis Type** is **Thermal** or **Coupled** and **Data to Map** is set to **Solid Temperatures**, the software creates structural **Node ID Table Temperature** loads.
- When the **Source Model Analysis Type** is **Flow** or **Coupled** and **Data to Map** is set to **Flow Values**, the software creates structural **Node ID Table Force** loads.

When the **Analysis Type** is **Axisymmetric Mapping**, the software creates structural **Node ID Table Temperature** loads.

You can create more than one solution at the same time, such as one for both Nastran and ANSYS. In that case, the mapping solver creates one set of loads that are shared between the solutions.

Why should I use it?

The automatic creation of mapping solutions for Nastran, ANSYS, and Abaqus simplifies the mapping of transient solutions.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Mapping	Thermal-Flow
NX Space Systems Thermal	Mapping	Thermal
NX Thermal and Flow	Mapping	Thermal-Flow
Axisymmetric Mapping	Thermal	

Where do I find it?

Application	Advanced Simulation
Prerequisite	Two FEM with models that are geometrically congruent
Simulation Navigator	Right-click the simulation file node → **New Solution**
Toolbar	**Advanced Simulation** toolbar → **Solution**
Menu	**Insert** → **Solution**
Location in dialog box	**Analysis Type** → **Mapping** or **Axisymmetric Mapping**

Mapping's element support

What is it?

The mapping solver now supports all elements which support temperature and pressure data for the following solvers:

- Nastran
- ANSYS
- Abaqus

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Mapping	Thermal-Flow
NX Space Systems Thermal	Mapping	Thermal
NX Thermal and Flow	Mapping	Thermal-Flow
Axisymmetric Mapping	Thermal	

Where do I find it?

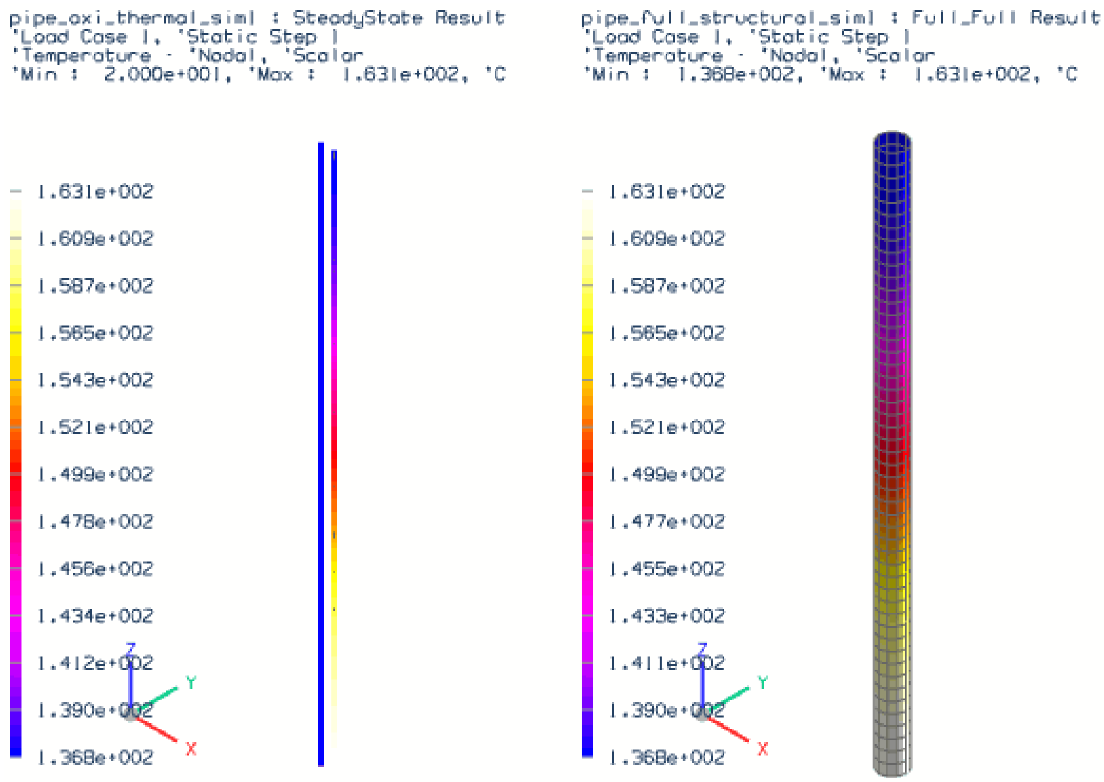
Application	Advanced Simulation
Prerequisite	Two FEM with models that are geometrically congruent
Simulation Navigator	Right-click the simulation file node → **New Solution**
Toolbar	**Advanced Simulation** toolbar → **Solution**
Menu	**Insert** → **Solution**
Location in dialog box	**Analysis Type** → **Mapping** or **Axisymmetric Mapping**

What is it?

You can now map temperature data from an axisymmetric model to a 3D model.

The following graphic shows:

- On the left, the temperature on 1D elements that model thermal analysis on an axisymmetric pipe.
- On the right, the same temperatures mapped on the shell elements of a full pipe.



Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Mapping	Thermal-Flow
NX Space Systems Thermal	Mapping	Thermal
NX Thermal and Flow	Mapping	Thermal-Flow
Axisymmetric Mapping	Thermal	

Where do I find it?

Application	Advanced Simulation
Prerequisite	Two FEM with models that are geometrically congruent

Simulation Navigator	Right-click the simulation file node → **New Solution**
Toolbar	**Advanced Simulation** toolbar → **Solution** 
Menu	**Insert** → **Solution**
Location in dialog box	**Analysis Type** → **Mapping** or **Axisymmetric Mapping**

Laminate Composites

Smart selection in draping

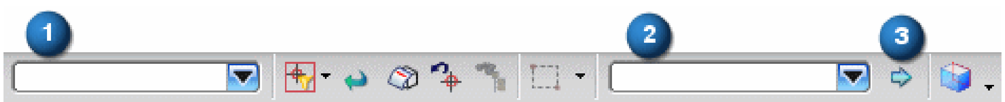
What is it?

Smart selection methods are now available in draping. Smart selection methods enable you to select entities according to specified criteria or entity relations.

Smart selection methods are available from the following dialog boxes:

- **Ply draping data**
- **Layup Offset**
- **Laminate Default Orientation**

When selecting entities, you can use the **Method** list on the Selection bar to select related entities.



The Selection Bar, showing (1) the Type Filter list, (2) the Method list, and (3) the Smart Selector Options button

Why should I use it?

Smart selection is most useful when working with complex geometry or meshes containing a large number of nodes and elements. If you use smart selection when selecting nodes, for example, you can easily select a large number of related nodes, either by their underlying polygon geometry features or by element tangency or feature angle. When working with models with complex geometry, you can use smart selection to quickly select all fillet faces, cylindrical faces, or sliver faces, as well as tangent or adjacent faces.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with one or more ply-based laminate layups defined
Simulation Navigator	• Right-click the ply node → **Edit** • Right-click the **Layup Offset** node → **Create a new user**

	defined layup offset rule • Right-click the **Material Orientation** node → **Create New Orientation**

Update icon in Simulation Navigator

What is it?

A red update icon now appears in the **Simulation Navigator** when either the layup or the ply needs to be updated.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with one or more ply-based laminate layups defined

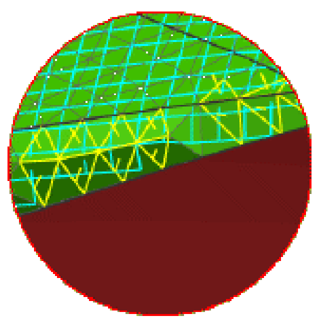
Draping solver feedback

What is it?

When you update layups or plies for draping, the **Information** window opens, and as necessary, displays the following warning messages:

- `WARNING — Bad continuity detected for ply X. Draping results could be wrong. Check element normals, slope continuity and mesh density.`

  In the graphics window, the regions with bad continuity between elements are highlighted. Element edges and the element normal are shown in yellow.

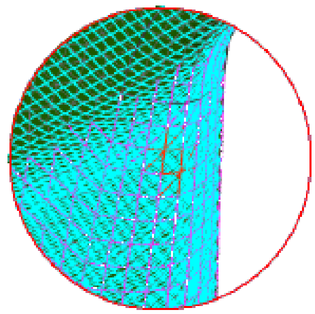
  

- `WARNING — Flat Pattern of plyX tears or overlaps Y times.`

  In the graphics window, the regions where elements tear or overlap are highlighted. Element edges are shown in red.

- `WARNING — Flat Pattern of plyX might tear or overlap Y times.`

  In the graphics window, the regions where elements might tear or overlap are highlighted. Element edges are shown in magenta.



The last two warnings are issued when the algorithm detects stretching in fibers, which could indicate that the ply cannot conform to the undevelopable faces. In practice, the ply would exhibit gapping or wrinkling on the mold.

Use one of the following remedial actions to improve the draping process:

- Use a different start point and draping direction.
- Add a splice in the ply.
- Introduce cut curves.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with one or more ply-based laminate layups defined

List draping results

What is it?

You can now export draping information for a given ply to a spreadsheet application or a comma separated values (CSV) file using the **List Draping Results** dialog box.

For a given ply, the created report contains the draping solver that was used, the starting and lock angles, and the number of elements.

For each element, the report contains:

- Element ID
- Shear angle
- Yarn angle
- Primary direction
- Secondary direction
- Normal direction

Why should I use it?

Use this functionality when you want to evaluate the validity of your draping. This is mainly a debugging tool.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with one or more ply-based laminate layups defined
Simulation Navigator	Right-click the ply node → **List Draping Results**

Envelope results for each element

What is it?

When you use the **Graphical Post Report** command, you can now view the worst ply value at each element for:

- **Ply Failure Index**
- **Bond Failure Index**
- **Ply Stress**
- **Ply Strain**
- **Ply Safety Margin**
- **Bond Safety Margin**

The new ply envelope nodes in the **Post Processing Navigator** have the same name as existing ply envelope nodes. In the existing nodes, you can view the values for each ply separately. In the new nodes, you can view the value from the ply with the worst result at each element.

In the following figure, the new ply envelope nodes are followed by 1 and the existing ones by 2 .

- Laminate Post Report 1
  - Ply Failure Index — Elemental 1
  - Ply Failure Index Bottom — Elemental
  - Ply Failure Index Top — Elemental
  - Ply Failure Index — Elemental 2
  - Bond Failure Index — Elemental 1
  - Bond Failure Index Bottom — Elemental
  - Bond Failure Index Top — Elemental
  - Bond Failure Index — Elemental 2
  - Ply Stress — Elemental 1
  - Ply Stress Bottom — Elemental
  - Ply Stress Top — Elemental

- Ply Stress — Elemental 2
- Ply Strain — Elemental 1
- Ply Strain Bottom — Elemental
- Ply Strain Top — Elemental
- Ply Strain — Elemental 2
- Ply Safety Margin — Elemental 1
- Ply Safety Margin Bottom — Elemental
- Ply Safety Margin Top — Elemental
- Ply Safety Margin — Elemental 2
- Bond Safety Margin — Elemental 1
- Bond Safety Margin Bottom — Elemental
- Bond Safety Margin Top — Elemental
- Bond Safety Margin — Elemental 2

Why should I use it?

The new ply envelope nodes allow you to assess the worst ply results without having to manually scan through all the ply results.

Where do I find it?

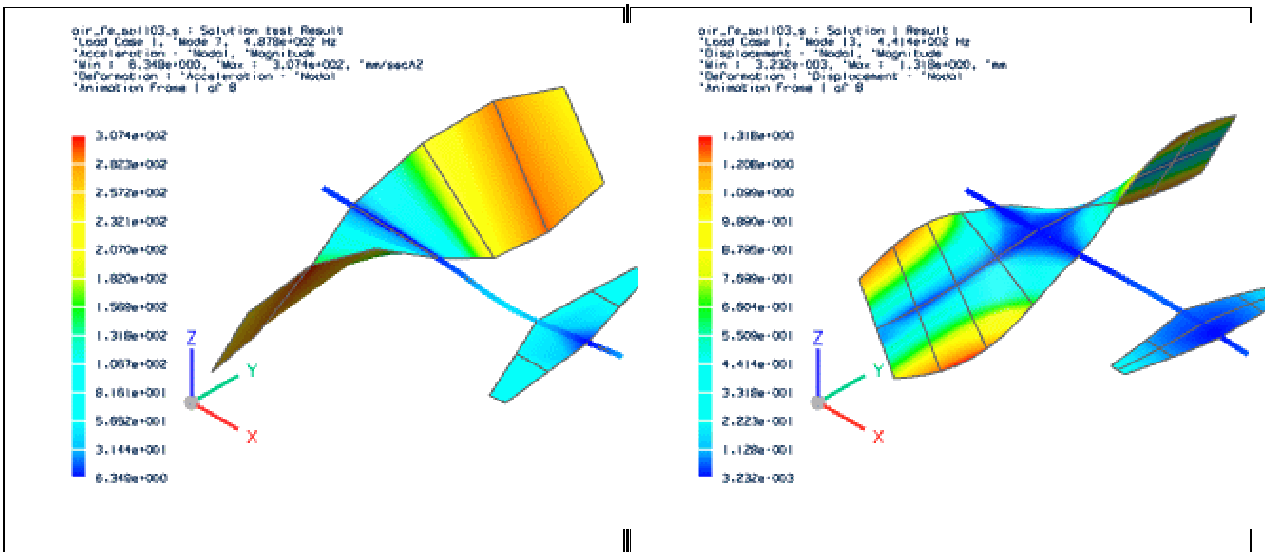
Application	Advanced Simulation
Prerequisite	A Simulation file with a laminate model in Nastran, ANSYS, or Abaqus solution
Toolbar	**Laminate → Graphical Post Report**
Menu	**Insert → Laminate → Graphical Post Report**
Location in **Post Processing Navigator**	Under the **Laminate Post Report** node

FE Model Correlation

What is it?

FE Model Correlation is a new Advanced Simulation process which you can use to quantify and visualize the level of agreement between two sets of modal results by comparing mode shapes and nodal responses.

FE Model Correlation can correlate modal results between structural finite element analysis and experimental test modal analysis results (test-analysis) or between two sets of structural finite element analysis results (analysis-analysis).



7th mode shape of the reference solution | **13th mode shape of the work solution**

You prepare the finite element model for structural analysis by defining the mesh and material properties. Then, you solve the following modal analysis solutions:

- NX or MSC Nastran: SEMODES 103
- ANSYS: Modal
- ABAQUS: Frequency Perturbation Substep

With FE Model Correlation, you create a solution with the new solver environment, **MODAL TEST DATA**, and the new solution type, **Test Solution**. You then import the experimental modal results and the test model from NX I-deas Universal (UNV) files into that solution. The test model is disjoint from the finite element model.

FE Model Correlation allows you to:

- Setup the correlation process.
- Align test solution with the analysis solution.
- Pair the test modes with the analysis modes.
- Obtain correlation metrics.

Why should I use it?

FE Model Correlation objectively compares mode shapes and nodal responses to assess the level of agreement between experiments and FE results in structural dynamics.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with modal analysis solution
Menu	**Insert→Correlation**

Import and export

Import changes for related elements that share a physical property table

What is it?

When you import related elements (an element type and its transitional element) that share the same physical property table, the software now places them in the same mesh in the **Simulation Navigator**. In previous releases, the software automatically created a separate mesh in the **Simulation Navigator** for every combination of physical property and element type.

For example, consider the following Nastran .dat file:

```
GRID,1,,0.,0.,0.
GRID,2,,1.,0.,0.
GRID,3,,1.,1.,0.
GRID,4,,0.,1.,0.
CQUAD4,1,1,1,2,3,4
CTRIA3,2,1,1,2,3
PSHELL,1,1,.1
MAT1,1,1.E10,.3
ENDDATA
```

Notice that both the CQUAD4 and CTRIA3 elements reference the same PSHELL bulk data entry.

Previously, if you imported this file into Advanced Simulation, the software created the following structure for the mesh data in the **Simulation Navigator**:

2D Collectors

PSHELL_1

CQUAD4_1

CQUAD3_2

Now, the software combines the CQUAD4 and CTRIA3 elements into the same mesh:

2D Collectors

PSHELL_1

CQUAD4_1

Where do I find it?

Application	Advanced Simulation
Menu	**File→Import→Simulation**

Option to import nodal temperatures as a node ID table

What is it?

For Nastran, Abaqus, and ANSYS models, the **Import Selective Loads as Field Data** customer default from previous releases now also controls whether nodal temperatures are imported as node ID tables. Whether you import nodal temperatures as loads or as initial conditions, if you select **Import Selective Loads as Field Data**, the software combines all nodal temperature boundary conditions into a single node ID table in the **Simulation Navigator**.

Previously, the **Import Selective Loads as Field Data** option only applied to forces and moments. Each imported nodal temperature boundary condition was displayed separately. This remains the default behavior.

Why should I use it?

When you import models with many nodal temperature boundary conditions, the **Import Selective Loads as Field Data** option improves performance and usability. The default method may result in slower import processing and require a large amount of scrolling to view all the boundary conditions in the **Simulation Navigator**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with NX Nastran, MSC Nastran, Abaqus, or ANSYS as the specified solver
Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Simulation→General→Environment** tab

Ability to export a subset of your model

What is it?

For Nastran, ANSYS, and Abaqus solutions, use the options in the new **Subset Export** group in the **Export Simulation** dialog box to selectively export portions of your model. You can use these options to export:

- Only the elements currently displayed in your model.
- Entities in selected groups.

Subset export based on the elements currently displayed

Use the new **Visible elements** option to export only the elements that are currently displayed in your model. This option replaces the **Visible meshes** option from previous releases. With the **Visible meshes** option, even if only a single element in a mesh was displayed, the software exported all the elements in that mesh.

Subset export based on selected groups

Use the new **Selected groups** option to export only the contents of a selected group. If you select multiple groups, the software exports all the entities in those groups as well as:

- Any loads and boundary conditions associated with the nodes and elements in the group.
- All nodes associated with the elements in that group, even if the nodes themselves are not members of the group.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file in which Nastran, ANSYS, or Abaqus is the specified solver
Menu	**File→Export→Simulation**

NX Open

Enhanced NX Open support in CAE

What is it?

Several JA-based NX Open APIs are now available that let you automate Advanced Simulation operations. The following new functions have been added:

NX Open class	New function added
BaseFEModel	Provides access to the **Update Finite Element Model** command.
SimSolution	Provides access to solve a solution or write a solver input file.
SimSolution	Provides access to complete edit capability for **SimSolution** properties as well as Solver Parameters.
NodeElementManage	Creates a **NodesRepositionBuilder** object that lets you efficiently reposition large numbers of nodes.

For details about each new function, see the *NX Open Reference Guide* for the language you are using (for example, the *NX Open C++ Reference Guide*). You can find the NX Open reference guides in the NX Help Library under **Automation→NX Open**.

NX Open support for reading and displaying solver results

What is it?

Using the NX Open interface, you can now write custom automation code for reading and displaying solver results in Advanced Simulation.

Several new NX Open classes have been added to the CAE NameSpace.

- ResultManager
- IResult
- ImportedResult
- ResultAccess
- ResultParameters
- SolutionResult

Use these general steps for working with these new classes:

1. Create a Result object from a solution or by opening a result file.
2. Create a ResultAccess object by providing a Result object and a ResultParameters object.

   ResultAccess holds a state that can be modified using a ResultParameters object. ResultParameters is a container of result properties that can be queried, modified, and set to a ResultAccess object.
3. Query file data from the Result object such as number of nodes, elements, load cases, and so on.
4. Access state data from the ResultAccess object such as minimum and maximum results values, min-max locations, and so on.

For details about each new class, see the *NX Open Reference Guide* for the language you are using (for example, the *NX Open C++ Reference Guide*). You can find the NX Open reference guides in the NX Help Library under **Automation→NX Open**.

NX Open support for manual creation of all 3D solid element types

What is it?

You can use the new **ElemDimTypeAnySolid** option in the **CAE::ElementCreateBuilder** NX Open class to create a 3D element of any solid topology supported by your solver. In previous versions, you could create only Penta(6) and Hexa(8) elements.

For details about the new option, see the *NX Open Reference Guide* for the language you are using (for example, the *NX Open C++ Reference Guide*). You can find the NX Open reference guides in the NX Help Library under **Automation→NX Open**.

Post Processing

Unaveraged results across feature angle boundaries

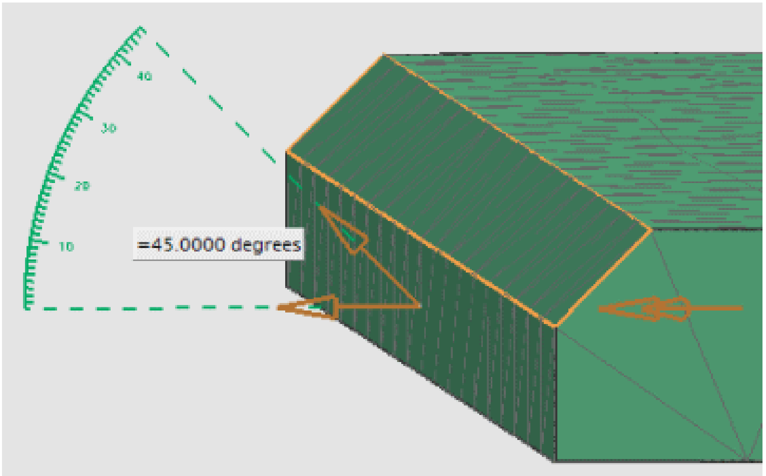
What is it?

When you analyze elemental or element-nodal results in post-processing and you use the **Averaged** results option, you can now choose not to average the results across nodes at feature angle boundaries in your model.

In the **Smooth Plot** dialog box, the **Averaged** option calculates a nodal average across all adjacent elements for elemental and element-nodal results. For this release, a new **Feature Angles** option has been added to the averaged-across options.

When the **Feature Angles** check box is not selected, the software leaves the results unaveraged across feature angles greater than the angle threshold you specify. For example, if you do not want the results averaged across feature angles of 45 degrees or greater, enter 45 in the angle threshold box.

The angle is measured at the free element faces, as shown in the following example.

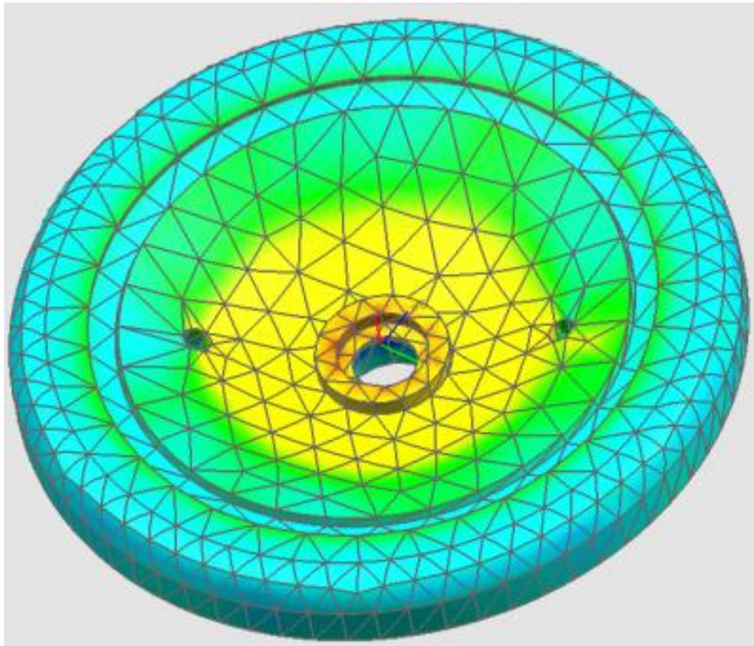


Note

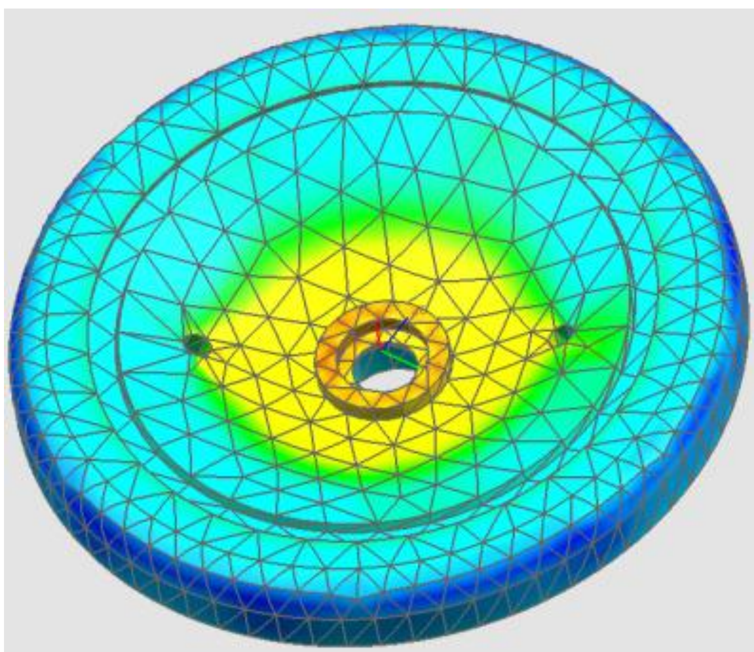
The **Include Internal Elements** option is unavailable when the **Feature Angles** check box is not selected.

Example

The following examples illustrate how the stress results remain unaveraged past the hole and slot features in this disk part.



Stress averaged across feature angles (default setting; Feature Angles check box selected)

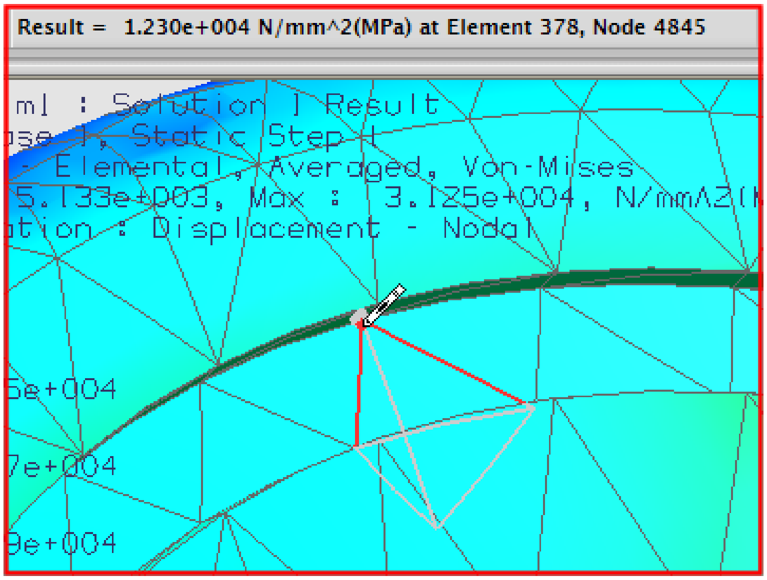


Stress unaveraged across feature angles greater than 45 degrees (Feature Angles check box not selected)

About the Identify command

You use the  **Identify** command to probe and display nodal information for the post view display. When you use the **Identify** command and the **Feature Angles** check box is not selected, you may see

multiple values on a single node, depending on the number of faces that share this node and the face you position the mouse over.



Where do I find it?

Application	Advanced Simulation
Prerequisite	Simulation file with elemental or element-nodal results. The **Include Internal Elements** option must be turned off.
Toolbar	**Post Processing**→**Post View** 
Location in dialog box	**Result** (**Color Display**)→**Averaged**→**Feature Angles**

Expanded results support in Post-Processing

What is it?

You can now display and analyze results in post-processing for several additional result types and element types.

NX Nastran results

- Contact gap distance
- Shell element thickness
- Gasket Pressure
- Gasket Closure
- Plastic Gasket Closure
- Gasket Status

- Gasket Yield Stress
- 2D axisymmetric contact results

NX Nastran elements

- Pyramid elements: CPYRAM
- Axisymmetric solid elements: CTRAX3, CQUADX4, CTRAX6, and CQUADX8

NX Response Simulation results

These Response Simulation contour results types are now supported correctly. In previous versions, all possible component results were displayed in the **Simulation Navigator** even if you had not requested certain components.

- Peak
- RMS
- Level Crossing

NX Laminates results

- Ply Failure Indices (ANSYS support added)
- Ply Strength Ratio
- Bond Strength Ratio

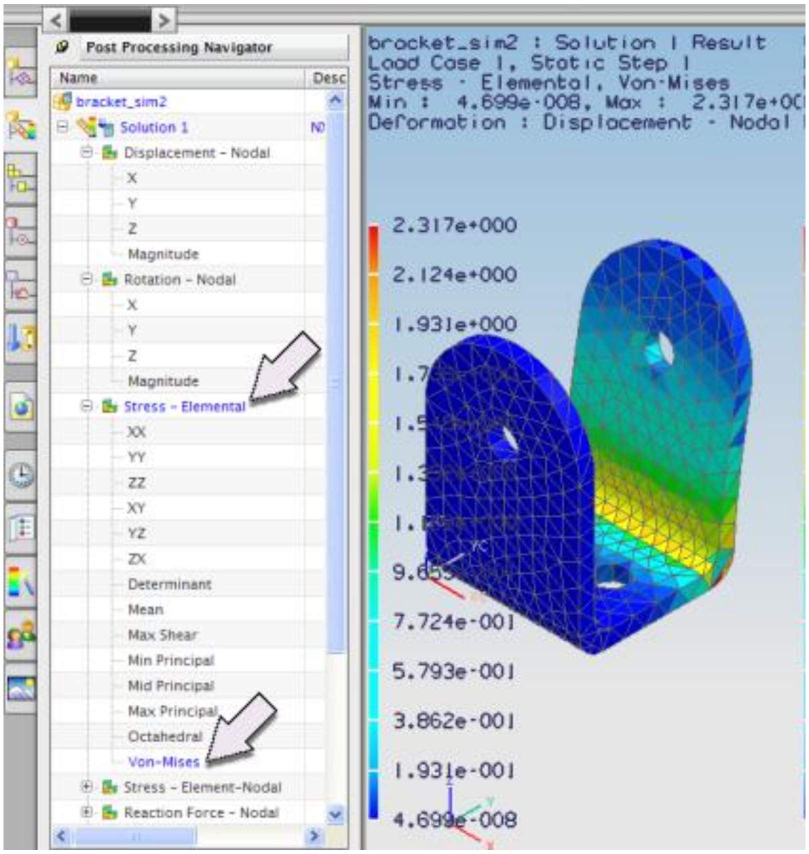
NX Flow results

Fluid density is now supported.

Context highlighting in Post-Processing Navigator

What is it?

The active results set that is plotted in the graphics window now displays in blue in the **Post-Processing Navigator**.



Von Mises stress result plotted in graphics window and displayed in blue in Post-Processing Navigator

If more than one results set is displayed, the Master post view is shown in blue in the navigator.

Why should I use it?

When you plot several results sets, the blue color in the **Post-Processing Navigator** can help you distinguish which set is displayed in the graphics window.

Restarting streamlines at the interface of a disjoint mesh

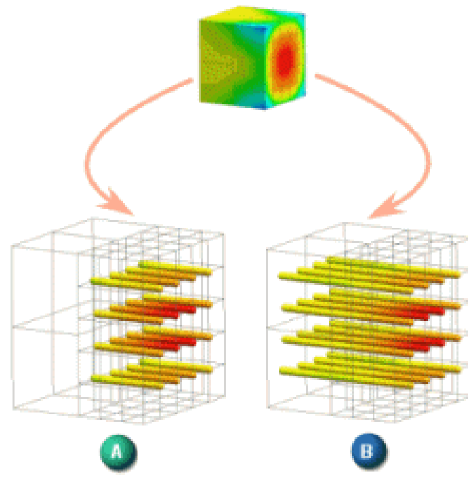
What is it?

If your model contains adjacent fluid meshes that share a common boundary, you can use the **Connect Disjoint Fluid Mesh** option in the **Create Solution** or **Edit Solution** dialog box to have the flow solver join them during the analysis. If you then create a streamline display of your velocity results for that model, you can use the new **Attempt Restart in Disjoint Mesh** option in the **Extraction Parameters** group of the **Seed Set** dialog box to control whether the streamline is continuous across the interface between the meshes. If you select **Attempt Restart in Disjoint Mesh**, the software creates a continuous streamline across the discontinuous meshes. However, restarting streamlines across discontinuous meshes can negatively impact the display performance. You should use caution when selecting this option.

Note

In the **Seed Set** dialog box, you must select the **Attempt Restart in Disjoint Mesh** option before you select any seed points.

The following graphic shows an example of a streamlines display created on a model that contains a disjoint fluid mesh. (A) shows the streamline display with the **Attempt Restart in Disjoint Mesh** cleared, while (B) shows the streamline display with the **Attempt Restart in Disjoint Mesh** selected.



Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved model with a disjoint fluid mesh, velocity results, and a displayed post view
Post-Processing Navigator	Right-click a post view→**Create Streamlines**

New results display options

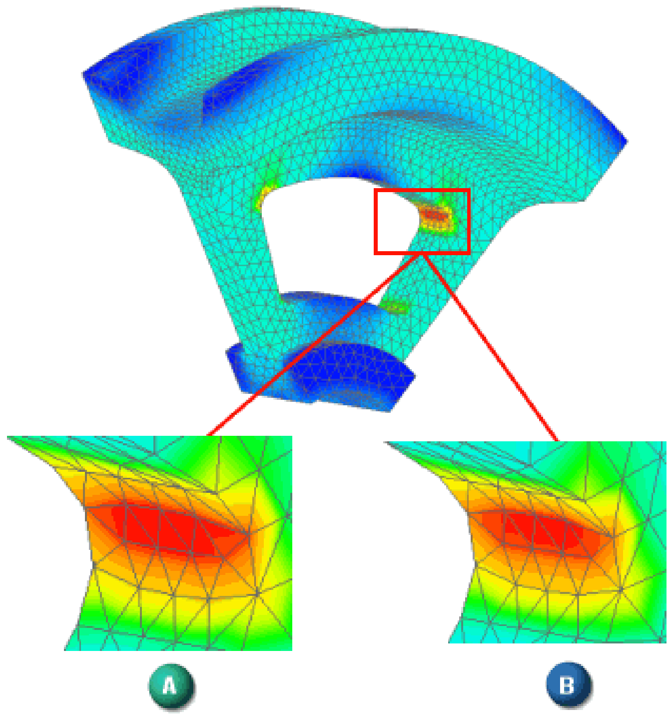
What is it?

Now, when you create a contour plot, marker plot, or streamline display of your results, new options in the **Select Results** dialog box give you greater control over how the software generates the display.

Option for averaging results for internal elements

If you select the **Averaged** option, you can use the new **Include Internal Elements** option to control whether the software includes the results for internal 3D elements in the averaging calculations. If you clear the **Include Internal Elements** check box, the software only includes elements that have faces on the model's surface in averaging calculations.

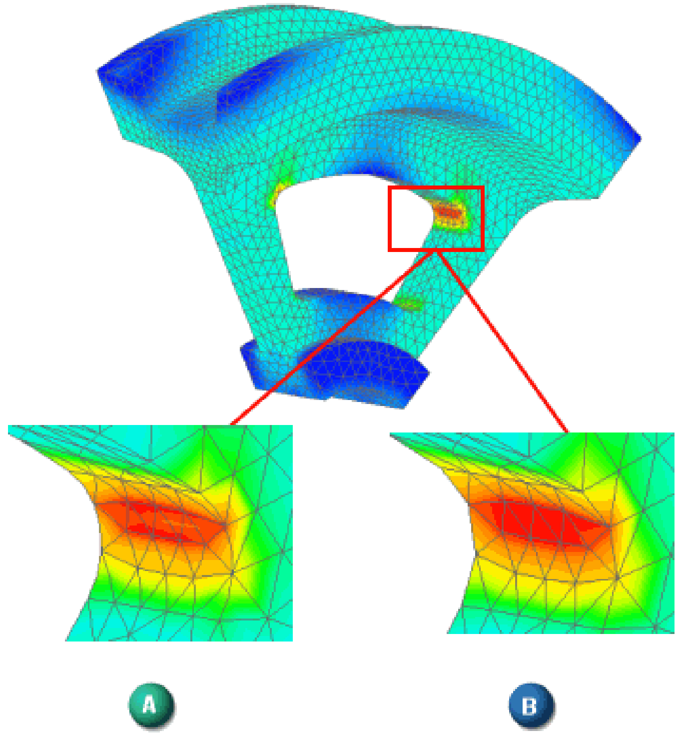
The following graphic shows contour plots of elemental stress-nodal results generated with the **Include Internal Elements** check box selected (A) and with the **Include Internal Elements** check box cleared (B).



Option for controlling the display of results at midside nodes

When your mesh includes parabolic elements, use the new **Include Midnodes** option to control whether the software displays results at the midside nodes.

The following graphic shows contour plots of elemental stress-nodal results generated with the **Include Midnodes** check box selected (A) and with the **Include Midnodes** check box cleared (B). The differences between the displays are very subtle for this model.



Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved model with results and a displayed post view
Menu	**Tools→Results→Post View**
Post-Processing Navigator	Right-click a post view→**Edit**

Nastran

Support for displaying Nastran strain curvature results

What is it?

In a Nastran analysis, when you include a **Strain** type of **Structural Output Request** with the **STRCUR** option from the **Plate Curvature** list selected, you can now display those results in post-processing. In previous releases, although you could request the output of strain curvature results, you could not display them.

Supported strain resultant components for 1D elements

In the **Post-Processing Navigator**, you can now create displays of the following strain resultant components for 1D elements:

- Axial Strain Exx
- Strain Curvature KXZ and KYZ
- Twist
- Shear Strain TXZ and TYZ

Supported strain resultant components for 2D elements

In the **Post-Processing Navigator**, you can now create displays of the following strain resultant components for 2D elements:

- Membrane Strain EXX, EYY, and EXY
- Strain Curvature KXX, KYY, and KXY
- Shear Strain TXZ and TYZ

Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved Nastran model with strain results
Toolbar	**Post Processing→Post View** 
Menu	**Tools→Results→Post View**
Post-Processing Navigator	Right-click the appropriate strain results→**Apply**

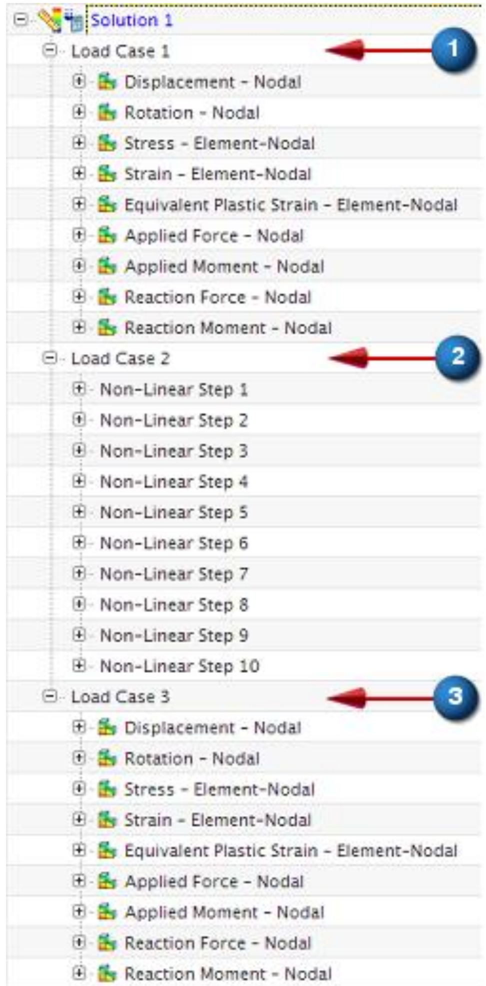
Abaqus

Ability to animate across load cases

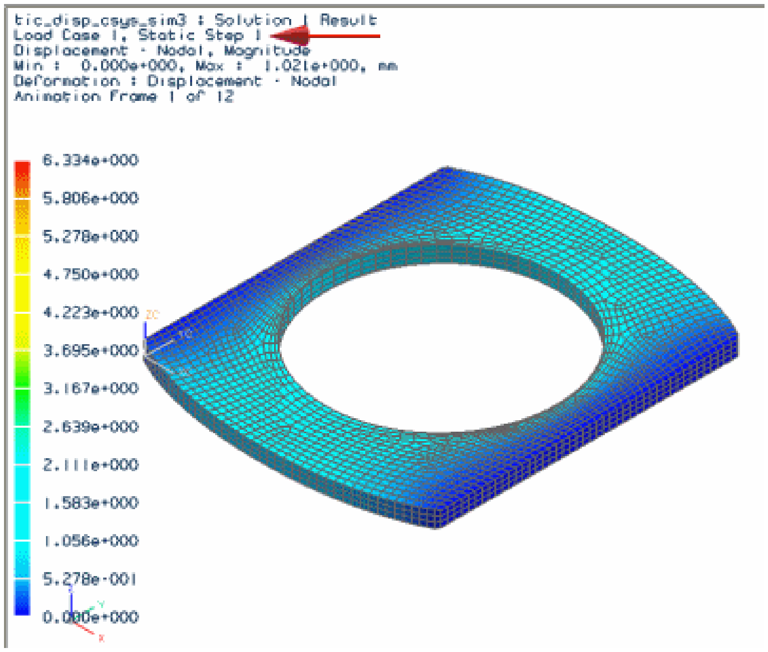
What is it?

For solutions that include multiple load cases, you can now animate the results across the load cases. The **Animation** dialog box includes two options: **Result** and **Iterations**. The **Iterations** option lets you specify a **Start Load Case**, **Start Iteration** (if the starting load case contains iterations), and **End Load Case**, as well as a **Step** increment to determine the number of frames in the animation.

For example, suppose you have three load cases (1, 2, and 3 in the picture below) and you want the animation to start at the first load case and end after the last load case.



In this example, you would choose **Load Case 1** as the **Start Load Case** and **Load Case 3** as the **End Load Case**. The following animated picture shows the result. The first and last load cases each have one static step and the second load case has 10 iterations. The **Step** option is set to **1**, which means the animation includes a frame for each step and iteration between the start and end load case. Therefore, the animation contains 12 frames. Note the load case and iteration number indicated by the arrow in the picture.



Note

In this example, all iterations in Load Case 2 were included automatically because it was not the starting or ending load case. You can specify a **Start Iteration** only if the load case that contains iterations is specified as either the **Start Load Case** or the **End Load Case**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation with Abaqus as the specified solver
Toolbar	**Post-Processing→Animation** 

Teamcenter Integration

Part attributes supported for FEM and Simulation files

What is it?

You can now assign **Part Name** and **Part Description** attributes to FEM and Simulation files in Advanced Simulation (in addition to **Part Number**, **Part Revision**, **Part Type**, and **Part Unit of Measure** attributes).

When you create a new FEM or Simulation file, you can now click the **Attributes** button, which opens the **Define Attributes** dialog box. There you can define the **Part Name** and **Part Description** attributes. You can also assign these attributes by selecting **Format→Database Attributes** to open the **Database Attributes** dialog box.

After initially defining these attributes, you can make changes to them until you save the part.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Standard**→**New** →**Simulation** tab→**Attributes**
Menu	**File**→**New**→**Simulation** tab→**Attributes**
	Format→**Database Attributes**
Simulation Navigator	Right-click the part file→**New FEM**→**Attributes**
	Right-click the FEM file→**New Simulation**→**Attributes**

Teamcenter file browser used when importing simulation results files

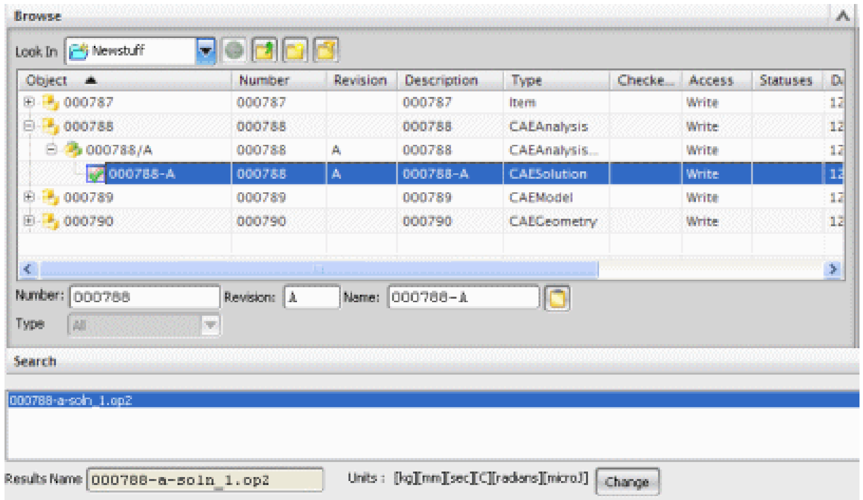
What is it?

You can now import results files associated with Simulation files managed in Teamcenter. In the **Simulation Navigator**, you can now right-click the Simulation file node and choose from two import options:

- **Import Local Results** — Lets you select a solver results file to import using the native operating system file browser.
- **Import Results file from Teamcenter** — Lets you select a solver results file to import using the **Import Results File from Teamcenter** dialog box. You can browse your Teamcenter database and select a Simulation file. The dialog box provides a view of each file's Teamcenter attributes, such as **Number**, **Revision**, **Description**, and so on. Results files stored with that Simulation file are displayed in the **Choose result file to import** list.

Also, in Advanced Simulation dialog boxes in which you select a simulation results file to import, the file system browser that appears is now the Teamcenter file browser.

For example, you can run a thermal solution to generate results for a temperature preload to a structural solution. In the structural solution, you choose **Teamcenter** as the **Source** when you define your temperature preload. The Teamcenter file browser appears, where you can browse your Teamcenter database and select the Simulation dataset that contains the results. The dialog box provides a view of each file's Teamcenter attributes, such as Number, Revision, Description, and so on. Results files stored with that Simulation dataset are displayed in the list.



Where do I find it?

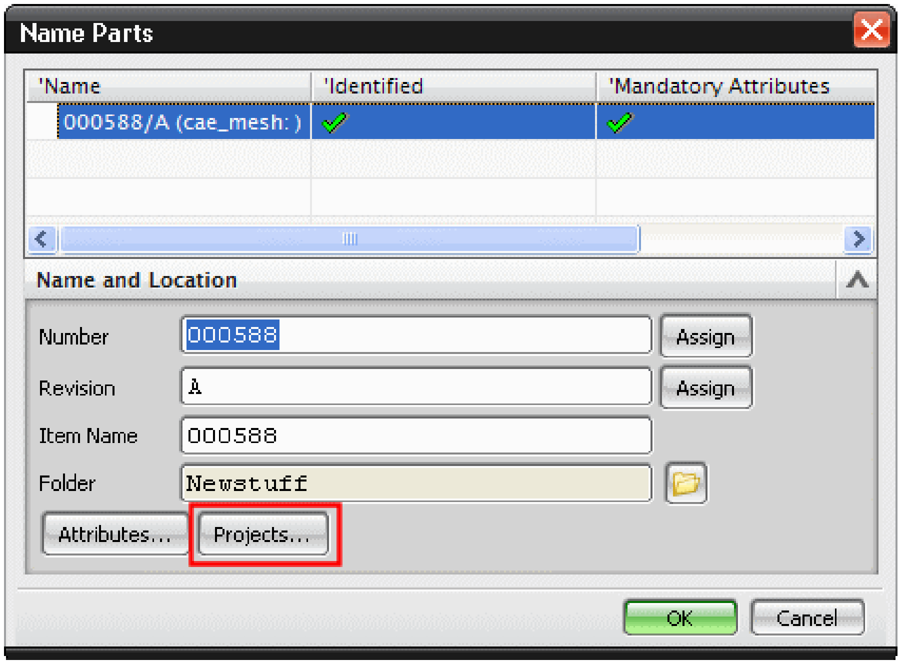
Application	Advanced Simulation
Menu	**File→Import→Results file from Teamcenter**
Simulation Navigator	Right-click the Simulation file node→**Import Results file from Teamcenter**

Assign a Teamcenter project to a CAE item

What is it?

In the Advanced Simulation application, you can now assign a Simulation file or FEM file to a Teamcenter project as you create the file.

A **Projects** button has been added to the **File New** dialog box, **Name Parts** dialog box, and **Save Part File As** dialog box.



The **Projects** button opens the **Assign to Project** dialog box, where you can select from a list of projects created in Teamcenter.

Motion Simulation

RecurDyn Solid 3D Contact

What is it?

A new type of **3D Contact** has been added for the RecurDyn solver. The **Solid** contact method generates a force between two links when they come into contact. Each link is represented as a single solid for contact purposes. This method solves much faster and is more accurate than the existing **Faceted** and **Fitted** contact methods.

You should use the **Solid** contact method for most contact situations. In general, the **Faceted** and **Fitted** methods are now useful only for backward compatibility with existing Motion mechanisms.

For more information, see 3D Contact for solid bodies in the Motion Simulation online Help.

Where do I find it?

Application	Motion Simulation
Toolbar	**Motion** toolbar→**3D Contact** 
Menu	**Insert**→**Connector**→**3D Contact**

Joint Wizard support for assembly constraints

What is it?

The Motion Simulation **Joint Wizard** now converts assembly constraints (as well as legacy mating conditions) to appropriate links and joints automatically. In previous releases, only mating conditions were supported.

The wizard creates the appropriate joint type based on the degrees of freedom in the components referenced by the assembly constraint. For example, if the assembly constraint removes five degrees of freedom (leaving one rotational degree of freedom), the **Joint Wizard** creates a revolute joint.

Translational DOF	Rotational DOF	Joint to create
0	0	Fixed
0	1	Revolute
1	0	Slider

Translational DOF	Rotational DOF	Joint to create
1	1	Cylindrical
2	1	Planar
0	3	Spherical or AtPoint
0	2	Universal
1	3	InLine
2	3	InPlane
3	0	Orientation
3	1	Parallel
3	2	Perpendicular

The wizard also infers the origin and orientation of the joint based on the geometry referenced by the assembly constraint. When you convert a joint from an assembly constraint, the wizard creates a smart point for the joint marker origin. This makes the joint position update automatically if the CAD model is modified.

The **Joint Wizard** runs automatically when you create a new Motion Simulation from an assembly that contains either mating conditions or assembly constraints.

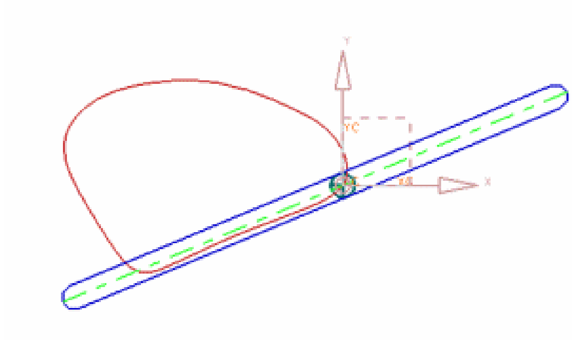
Where do I find it?

Application	Motion Simulation
Prerequisite	Assembly that contains mating conditions or assembly constraints

Driver for Point on Curve constraint

What is it?

With the RecurDyn solver, you can now create a motion driver for use with a **Point on Curve** constraint. Use the **Driver** dialog box to create a **Constant**, **Harmonic**, or **Function** driver (**Displacement** and **Velocity** data types only) and assign it to the **Point on Curve** constraint. This ability lets you drive the motion of the point on curve tangential to the curve.



Also, new **Customer Defaults** settings have been added to control how NX communicates the curve representation to the RecurDyn solver. In most cases, the default setting is recommended. However, if you have problems solving the mechanism, you can try changing the new **Curve Parameterization Method** in **Customer Defaults**, as described in About the Point on Curve method in Customer Defaults.

Where do I find it?

Application	Motion Simulation
Prerequisite	**Normal Run** solution
Toolbar	**Motion** toolbar→**Driver** 
Menu	**Insert**→**Driver**

Quasi Static Articulation option for Adams/Solver

What is it?

When using Adams/Solver and running an **Articulation** solution, you can now choose the algorithm for Adams/Solver to use: **Quasi Static** or **Dynamic**.

- **Quasi Static** — (Default) Use this method for most **Articulation** simulations.
- **Dynamic** — Use this method when your mechanism contains nonlinear attributes such as many bushings and/or high damping coefficients.

Note

Generally, with this type of nonlinear mechanism, you should use a **Normal Run** solution instead of **Articulation**. However, if you must use **Articulation**, use this **Dynamic** option.

In previous releases, the solver selected the method automatically, but this caused problems when solving some mechanisms.

For detailed information about each algorithm, see your Adams/Solver documentation.

To change the **Articulation** algorithm:

1. From the **File** menu, choose **Utilities→Customer Defaults**.
2. In the **Customer Defaults** dialog box, choose **Motion→Analysis**.
3. On the **Adams** page, select the appropriate option.

Chapter 9: NX Analysis

Space Finder

What is it?

The **Space Finder** command enhancement lets you:

- Set a user defined precision for inner volume calculation.
- Send the results to the **Information** window.

Why should I use it?

Use a user defined precision when other precision options do not produce results with the accuracy you expect.

Where do I find it?

Menu	**Analysis** → **Space Finder**

Chapter 10: Product Template Studio

Product Template Studio enhancements

What is it?

The **Product Template Studio** application now has the ability to:

- Create templates from FEA context.
- Create templates from motion studies context.
- Clone drawings when a template is executed.

The following objects are now supported in a template:

- Motion files and objects such as:
  - Joints, links, dampers, couplers
  - Forces
  - Simulations and solutions
  - Animation controls
  - Packaging objects
  - Results
  - Graphs
- FEM objects such as:
  - Loads
  - Constraints and boundary conditions
  - Simulations and solutions
  - Results

Why should I use it?

These enhancements give you the ability to capture and reuse robust practices in motion studies and FEA analysis.

Where do I find it?

Application	**Product Template studio**
Prerequisite	You must open a model and select a feature in the **Model Explorer** window.
Location in dialog box	**Attributes** tab

Knowledge Fusion Visual Rules enhancements

What is it?

The Product Template Studio application now allows you to add knowledge fusion visual rules to product templates. These rules are decision logics in the following categories:

- Fetch — Fetch objects such as parts, features, bodies, reference sets and so on.
- Filter — Filter objects such as parts, features, expressions, geometry and so on.
- Construct — Create conditional components and decision logic loops.
- Act — Adds or removes objects such as components, attributes, expressions, layers and so on.
- Result — Sends the results to the information window or an HTML file which you can author.

Why should I use it?

Use the knowledge fusion visual rules to visually add knowledge fusion logic to new and existing product templates.

Where do I find it?

Application	**Product Template Studio**
Prerequisite	You must open a model.
Location in dialog box	User interaction panel

Chapter 11: Flexible Printed Circuit Design

Rebend enhancement

What is it?

When you use the **Rebend** command, you can specify a stationary face during the rebending operation.

This command works the same as the **Rebend** command in NX Sheet Metal. For more information, see NX Sheet Metal — Rebend enhancement.

Where do I find it?

Application	Flexible Printed Circuit Design, NX Sheet Metal
Toolbar	**Flexible Printed Circuit Design→Rebend** **NX Sheet Metal→Rebend**
Menu	**Insert→Form→Rebend**

Chapter 12: Enhancements in NX 6.0.x Maintenance Releases

Gateway

Issue Navigator

What is it?

Use the **Issue Navigator** to capture, track, and close issues that arise during your design process. Issues can be design problems, workflow problems, or anything else you want to track in NX.

Before you can use the navigator, you must do the following. See the NX 6.0.3 Help for more information.

- Make sure the Teamcenter Community site where your issues lists are located has at least one existing issue list. If not, create the issue lists that you want to use.
- Set a customer default to specify the URL of the Teamcenter Community site where your issues lists are located. You should also specify the other defaults for the **Issue Navigator**, which define navigator columns, filter conditions (if any), email notification for the person assigned to fix the issue, and attachment download behavior for native NX.
- Select the **Issue Navigator** in the **Start** menu.

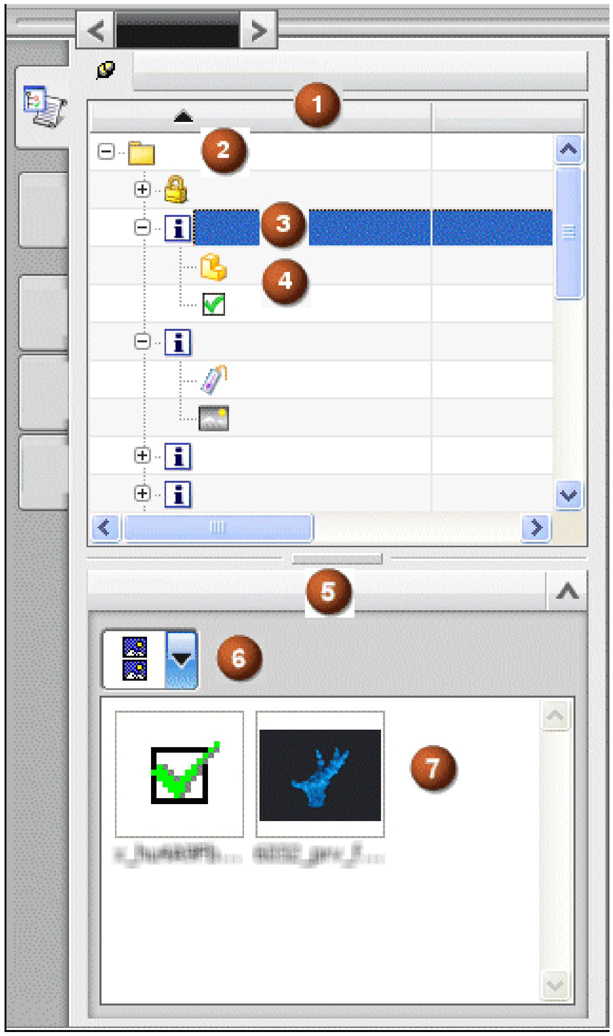
Consider the following behavior when you create or modify an issue:

- You must log into the site where your issue lists are located before you can work on any issues in native NX, or in Teamcenter Integration when Teamcenter Community is the site. To log in, right-click the background or a column heading in the main panel of the navigator.

  Note

  In NX 6.0.3, the site must be Teamcenter Community.

- You can attach part files, bookmarks, text documents, and other reference information to an issue.
- You can assign the issue to the person who is responsible for fixing it. Optionally, when you set up the navigator, you can specify that it send e-mail notification to the assignee when the issue is created, modified, or closed.



Simplified representation of the Issue Navigator

1	**Issue Navigator** main panel
2	Issue list
3	Issue In the figure above, an issue node is selected.
4	Attachments — the icon shows the type of attachment. In the figure above, the attachments for the selected issue are a part and a validation log.
5	**Details** panel

6	View types — specifies how the information in the details list (item 7) is shown. In the figure above, the **Thumbnails** view type is used.
7	Details list — displays the details of the selected issue or issue list, in accordance with the view type. In the example, the details list shows thumbnails for the attachments.

The server generates a unique ID for each issue.

You can group or filter issues in the **Issue Navigator** with the **Configure View** dialog box. Grouping issues makes them appear in the order you want, for example, based on priority. You can also create custom filtered views of the **Issue Navigator** that show only the issues that interest you.

Closed issues are read-only, and have a lock icon .

Why should I use it?

The **Issue Navigator** lets you manage user-defined issues and their supporting files in a single location in NX.

Where do I find it?

Menu	**Start→All Applications→Issue Navigator**
Resource Bar	**Issue Navigator**

Exact lightweight geometry and refile_part utility changes

What is it?

The lightweight (faceted) representation format has been enhanced to contain exact surface geometry information for faces with analytic surface geometry such as faces with planar, cylindrical, spherical, or toroidal surface geometry.

The software uses the exact geometry information while performing certain operations on faces, edges, and vertices of lightweight bodies. This information enables the software to perform the operations on lightweight bodies with the same accuracy as on solids. Examples of operations where exact geometry information is used include **Move Component** and many types of measurement.

If the exact geometry in the part is created or updated in NX 6.0.2 or later, then the exact geometry information will be included in the lightweight representation and used by NX where possible. For parts with geometry last modified prior to NX 6.0.2, you must regenerate the lightweight representations to benefit from the improvements.

The refile_part and ugmanager_refile utilities have been enhanced to facilitate the regeneration.

Switch	Description
regen_lw	Regenerates all lightweight representations in the part, in order to take advantage of NX 6.0.2 enhancements to the lightweight format, such as the embedding of exact surface geometry definitions for faces with analytic geometry.
regen_lw_def_tol	Regenerates all lightweight bodies using the current default faceting tolerance values. **Note** In Teamcenter Integration, regen_lw_def_tol must be used in conjunction with regen_lw to take effect.

Why should I use it?

These enhancements enable you to get precise results in some important situations where you would previously have gotten approximate results due to the faceted nature of the representations.

Where do I find it?

Lightweight representations created or edited in NX 6.0.2 automatically use exact lightweight geometry for analytic faces.

To update other parts to take advantage of the enhanced lightweight representation format, you can run the refile_part utility (in native NX) or the ugmanager_refile utility (in Teamcenter Integration), using the new switches, from the command line of your operating system.

See the *Utilities and File Management* Help and the *Teamcenter Integration for NX* Help for more information.

Specify geometry layer in JT files

What is it?

You can now set customer defaults to specify the layer on which to locate model geometry objects when you open a JT file in NX. Geometry objects include solid bodies, sheet bodies, and faceted bodies.

If you want the geometry to be placed on the work layer, select the **Use Work Layer** check box.

If you want the geometry objects to be placed on a different layer, clear the **Use Work Layer** check box, and specify the layer you want in the **Model Geometry Layer** box.

Why should I use it?

The ability to control the geometry placement layer in JT files helps you to:

- Follow your company standards for geometry placement on layers.
- Place geometry objects on the layer that best suits your design intent.

Where do I find it?

Menu	**File**→**Utilities**→**Customer Defaults**
Location in dialog box	**Gateway**→**Extras**→**JT Files** tab

Move Object enhancements

What is it?

The **Move Object** dialog box has the following new options:

Layer Option list

- **Work** moves or copies the selected objects in the current work layer.
- **Original** moves or copies the selected objects on their original layer.
- **As Specified** moves or copies the selected objects to the specified layer.

Layer box

Available only when **Layer Option** is set to **As Specified**.

Lets you specify the layer to which the selected objects are to be moved or copied.

Associative

Creates an associative Move Object feature.

Note

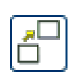
The **Associative** option is not available in Sketcher and Drafting applications.

Why should I use it?

Use the **Associative** option to create an associative Move Object feature.

The layer options let you specify the layer to which the selected objects are to be moved or copied.

Where do I find it?

Application	Gateway
Toolbar	**Standard**→**Move Object** 
Menu	**Edit**→**Move Object**
Location in dialog box	**Result** group→**Layer Option**/**Layer** **Settings** group→**Associative**

Modeling

Resize Datum Plane

What is it?

Use the **Resize Datum Plane** command to change the width and length of an existing datum plane.

Why should I use it?

You can change the size of a datum plane, even in large models, without updating the model. Although you can also resize a datum plane with the **Edit with Rollback** command, doing so requires an update of the model.

Where do I find it?

Application	Modeling
Menu	**Edit→Feature→Resize Datum Plane**
Graphics window	Right-click an existing datum plane→**Resize Datum Plane**

Convert to Linked Body

What is it?

Converts an unparameterized feature to a Linked Body feature. The Linked Body feature is without parents and appears in the Part Navigator as a broken link.

Any dependencies that may exist for the body, edges, and faces of the unparameterized feature are transferred to the corresponding body, edges, and faces of the converted Linked Body feature.

Why should I use it?

You can use this command any time you need to replace the body of an unparameterized feature with another body. You can edit the Linked Body to specify a new parent body of the part, and then use the **Replacement Assistant** to maintain associativity with downstream data.

This command may also be useful in avoiding phased migration with traditional migration methods. You can migrate all parts of an assembly at the same time, including Assembly and Drafting files.

Where do I find it?

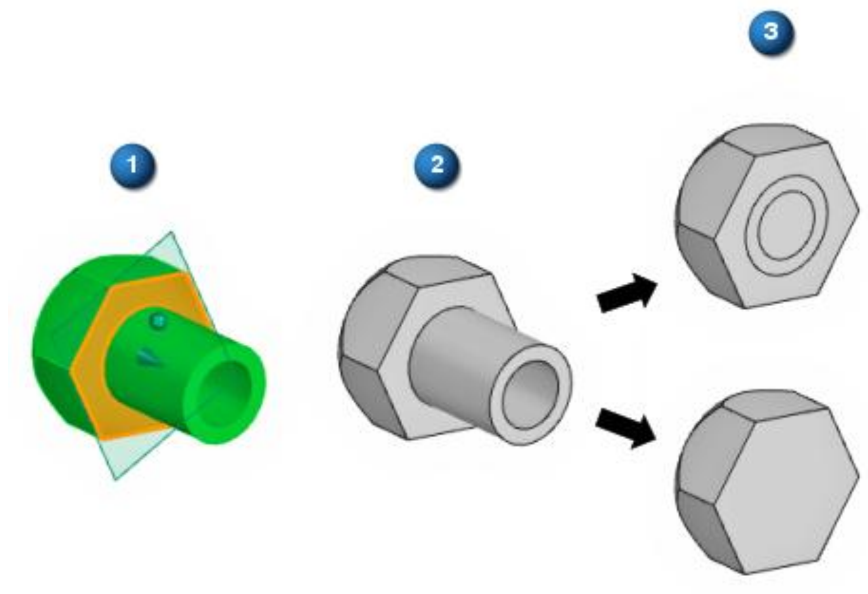
Application	Modeling
Part Navigator	Right-click an unparameterized feature→**Convert to Linked Body**

Retaining imprinted edges in a Split Body

What is it?

Keep Imprinted Edges is a new option for the **Split Body** command that lets you retain the edges that mark the intersection between the split bodies. The Advanced Simulation application uses the edges to automatically create glue coincident mesh mating conditions between the bodies.

1 The selected solid body in the figure is the target in a **Split Body** operation. A highlighted face plane is the tool that will split the body.



2 The result is two solid bodies adjoining each other.

3 If **Keep Imprinted Edges** was selected during the **Split Body** operation, hiding the cylinder solid body reveals the imprinted edges (top image).

If **Keep Imprinted Edges** was not selected, imprinted edges are not present when you hide the cylinder (bottom image).

Note

An existing edge can be considered an imprinted edge if it forms part of the intersection between faces from the target and the tool.

When working in Advanced Simulation with an idealized part active, the **Split Body** option **Keep Imprinted Edges** becomes **Auto Create Mesh Mating Conditions**.

Why should I use it?

Use this option to create edges between the bodies generated from a **Split Body** command, which are used in Advanced Simulation to automatically create mesh mating conditions. Mesh mating conditions

ensure that meshes are continuous from one body to the other. Previously, you had to manually create mesh mating conditions between any bodies created by the **Split Body** command.

Where do I find it?

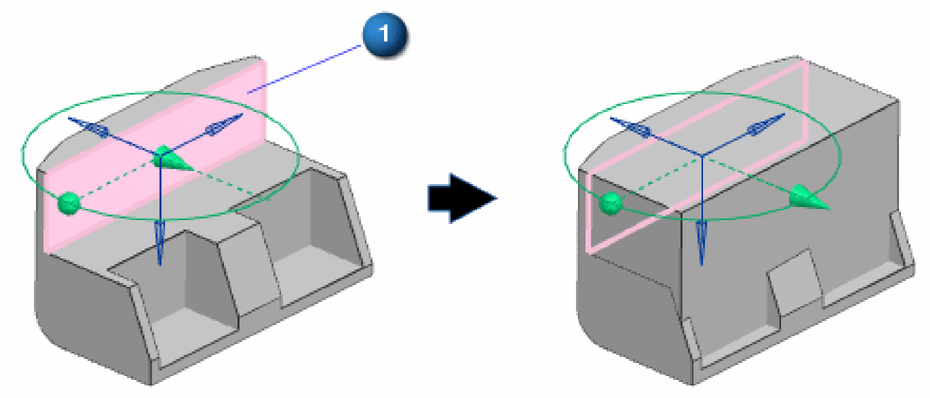
Application	Modeling, Advanced Simulation
Prerequisite	When working in Advanced Simulation, the idealized part must be active
Location in dialog box	(From Modeling) **Settings** group→**Keep Imprinted Edges** (From Advanced Simulation) **Settings** group→**Auto Create Mesh Mating Conditions**
Menu	**Insert**→**Trim**→**Split Body**
Toolbar	**Modeling**→**Insert**→**Trim**→**Split Body**

Synchronous Technology face overflow options

What is it?

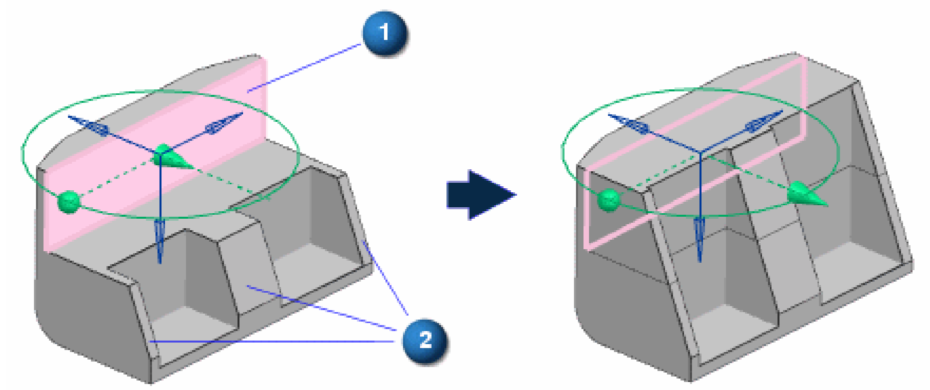
Synchronous Technology commands now include options to let you control overflow characteristics of faces you move, and how they interact with other faces.

Extend Change Face Dragging the selected face extends it into or moves it past other faces it encounters.



1 Selected face

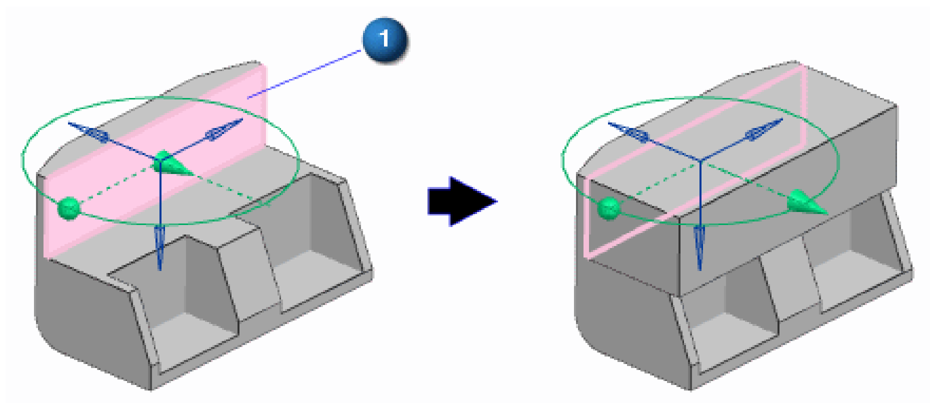
Extend Incident Face Dragging the selected face extends it until it meets a stationary, incident face. The selected face then ceases to extend, and the stationary face extends instead.



2 Incident faces

Extend Cap Face

Dragging the selected face past an overhanging edge causes it to overflow and cap itself (the bottom of the change face in the figure below).



Automatic

Dragging the selected face causes either the selected face or an incident face to extend, depending on which outcome would result in the least amount of change to volume and area.

Why should I use it?

Use these options to control how a change face overflows stationary or incident faces in solid bodies.

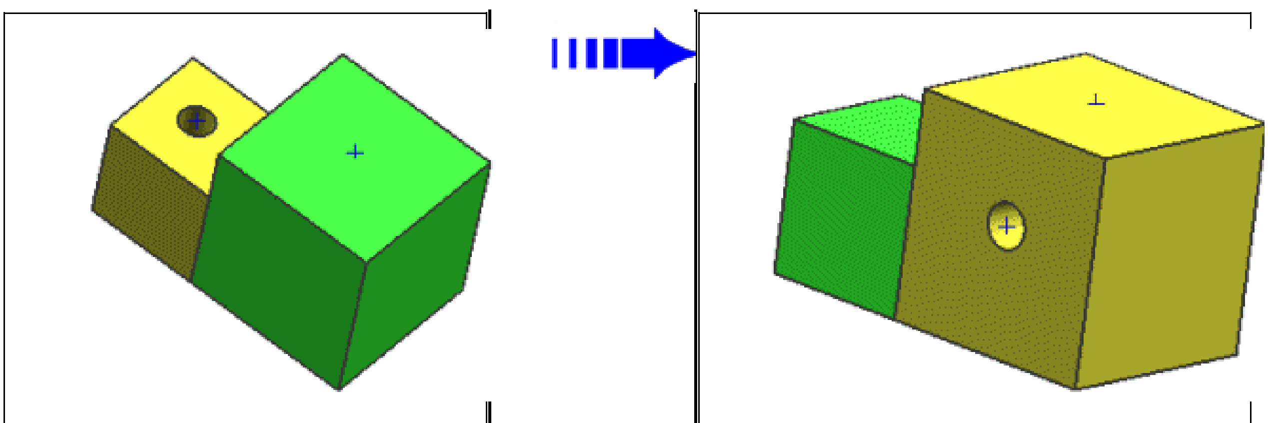
Where do I find it?

Application	Modeling
Location in dialog box	**Settings** group→**Overflow Behavior**
Menu	**Insert**→**Synchronous Modeling**→**Move Face**/ **Offset Region**/ **Replace Face**/ **Make Coplanar**/ **Make Coaxial**/ **Make Tangent**/ **Make Perpendicular**/ **Make Parallel**/ **Linear Dimension**/ **Angular Dimension**/ **Radial Dimension**
Toolbar	**Synchronous Modeling**

Synchronize Views in Replace Feature

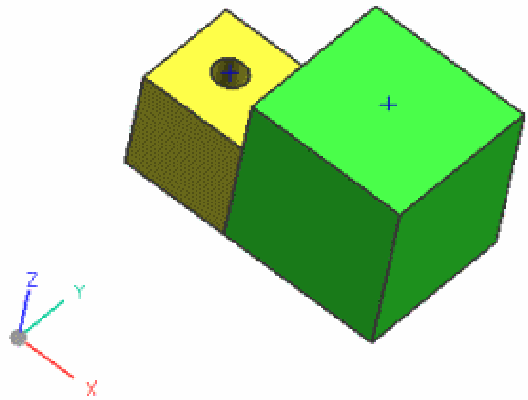
What is it?

Synchronize Views is a new option in the **Replace Feature** command. This options synchronizes the current and replacement views when you replace a feature. When you rotate, pan, zoom, or apply rendering styles in one view, NX automatically synchronizes the other view to match these operations.



Before replacement **After replacement**

The following animation demonstrates the automatic synchronization of the current view and the replacement view as you replace the hole feature from one block to the other.



A split screen appears only when:

1. The selected feature to replace has downstream dependents that appear in the **List** subgroup under the **Mapping** group.
2. You select any of the displayed dependents as reference from the **List** subgroup.

Why should I use it?

Use this option to locate objects that you want to replace in one view, while NX automatically tracks your movements toward the same location in the other view. This saves mouse clicks and reduces the need to repeat the same view manipulations in the other view.

Where do I find it?

Application	Modeling
Prerequisite	You must select a dependant reference of the selected feature to replace from the **List** subgroup in the **Mapping** group.
Toolbar	**Edit Feature→Replace Feature** 
Menu	**Edit→Feature→Replace**
Location in dialog box	**Settings** group→**Synchronize Views**

Assemblies

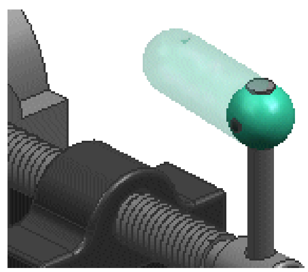
Motion envelopes

What is it?

The **Motion Envelope** function, which is used to create a volume of motion for components, has the following enhancements:

- New underlying swept volume generator technology that creates better motion envelopes more quickly. This technology is also used by Teamcenter Visualization.
- A simpler user interface for defining an envelope accuracy in the **Custom** quality option. Beginning in NX 6.0.2, when you set **Quality** to **Custom**, the quality is defined using a single **Envelope Tolerance** option, replacing the multiple, less intuitive options of the previous releases.

  Use the **Envelope Tolerance** option to specify the maximum distance between the theoretical and actual motion envelopes. Smaller tolerances produce more accurate envelopes, but require more time and memory.



Motion envelope of a vise handle ball

The new swept volume generator has the following requirements:

- A 3D graphics adapter with 24-bit depth buffer or better (Required)
- A system with at least 2GB of RAM (Recommended)

Note

The generation of high quality envelopes for complex parts may use a large amount of memory for some motion definitions. If you find that you are unable to generate motion envelopes due to virtual memory or RAM limits on your computer, you could try using a lower quality setting for the motion envelope, or restart NX so the largest amount of memory is available. Alternatively, try to create the envelope using a computer with more memory.

The new swept volume generator can make use of multiple CPUs or cores. However, the benefit of having more than two cores decreases rapidly, because most of the work is done by the graphics adapter.

Why should I use it?

Advantages of creating motion envelopes in NX 6.0.2, compared to earlier releases, include the following:

- Accuracy — the motion envelope is tighter at all quality levels: low, medium, high, and custom.
- Performance — the generation of a motion envelope of any quality is much faster.
- Usability — Custom envelope tolerances are much easier to understand.

Where do I find it?

Application	**Assemblies**
Prerequisite	You must be in the assembly sequencing environment, and your assembly sequence must include one or more motion steps.
Toolbar	**Assembly Sequencing and Motion→Motion Envelope** 
Menu	**Tools→Motion Envelope**

Variable positioning and Fix constraints

What is it?

When variable positioning is used on a component that includes Fix assembly constraints, the inherited version of each Fix constraint is now a Bond constraint.

When you open an assembly that includes Fix constraints inherited by variable positioning applied in an earlier release of NX, the higher-level inherited Fix constraints are converted to Bond constraints.

Why should I use it?

Inheriting a Fix constraint to a higher-level Fix constraint can cause undesirable behavior such as preventing movement of the fixed component's parents in higher-level assemblies. Converting the Fix constraint to a Bond constraint at higher levels preserves some of the behavior of the Fix constraint on the component. The higher-level Bond constraint connects the fixed component to its parent, which lets the component and parent move as a pair, but restricts independent movement of the component.

Where do I find it?

Application	**Assemblies**
Prerequisite	You can only use variable positioning on a component that has at least two assembly levels above it. This means that the component must have at least one parent that has a parent of its own. **Note** The lowest level can be a subassembly if you do not select any of its components.
Shortcut menu	**Assembly Navigator**→right-click a component node→**Override Position**

Drafting

Single Sided arrowhead terminators

What is it?

You can now select single sided arrowhead terminators on annotation with leaders. There are four new arrowhead types:

-  — **Top Open Arrow**
-  — **Bottom Open Arrow**
-  — **Top Closed Arrow**
-  — **Bottom Closed Arrow**

Why should I use it?

Use the single sided arrowhead terminators to maintain compliancy with ESKD (Russian) standards or to use other annotations (for example, Weld Symbols) that use half arrowheads.

Where do I find it?

Application	Drafting and PMI
Prerequisite	The single sided arrowhead options are only available in annotation command dialog boxes with groups.
Location in dialog box	**Leader** group→**Style** sub-group.

ESKD (Russian) standard thread display

What is it?

You can select an option to display threads on a drawing in compliance with the ESKD (Russian) standard.

Why should I use it?

Use the ESKD thread option to maintain compliancy with the ESKD (Russian) standards.

Where do I find it?

Application	Drafting
Location in dialog box	**View Style/View Preferences→Threads** **Customer Defaults→Drafting→General→Customize Drafting Standard→View→Threads** tab→**Thread Standard**

Restrict crosshatch angle

What is it?

The **Assembly Crosshatching** and **Restrict Crosshatch to +/- 45 degrees** options enable you to restrict assembly crosshatching angles to plus or minus 45 degrees in section views.

Why should I use it?

Use the restricted crosshatch angle to maintain compliancy with the ESKD (Russian) standards.

Where do I find it?

Application	Drafting
Location in dialog box	**Preferences**/**Style**→**View**→**Section** tab **Customer Defaults**→**Drafting**→**View**→**Hatching** tab

ESKD (Russian) weld symbols

What is it?

New standard for weld symbols that includes additional settings and symbol types required by the ESKD (Russian) standards. The additional settings available for ESKD weld symbols are:

- **Weld Line Gap**
- **Arrowhead Type**

> **Note**
>
> Color, font, and width settings are available for all weld standards.

The additional symbols available for the ESKD weld symbols are:

- **Flush Weld**
- **Machining for Graded Junction**
- **Intermittent Weld**
- **Weld Along Closed Contour**
- **Weld Along Open Contour**

New symbols to meet the ESKD (Russian) standards have also been added for:

- **Staggered Weld**
- **Field Weld**

Where do I find it?

To set the default standard for weld symbols

Application	Drafting or PMI
Menu	**File**→**Utilities**→**Customer Defaults**→
Location in dialog box	**Drafting**→**General**→**Standard**→**Customize Standard**→**Other Symbols** →**Weld** tab→**Standards**

GB (China) weld symbols

What is it?

GB is a new standard for weld symbols that includes additional settings and symbol types required by the GB (China) Standard. The additional symbol available for GB weld symbols is:

- **Trilateral Weld**

Where do I find it?

To set the default standard for weld symbols

Application	Drafting or PMI
Menu	**File**→**Utilities**→**Customer Defaults**
Location in dialog box	**Drafting**→**General**→**Standard**→**Customize Standard**→**Other Symbols**→ **Weld** tab→**Standards**

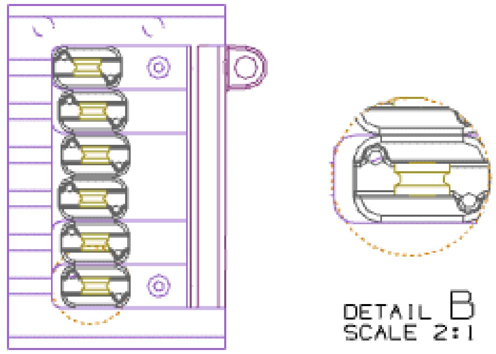
Detail View color, font, and width

What is it?

You can control the color, font and width settings for:

- Detail view boundary lines.
- Detail view labels on a parent boundary line.

The options to do this are available from Customer Defaults, Preferences, and Style.



Detail View with dotted, orange colored boundary and black text

Why should I use it?

You can use these settings to control how detail views and labels appear on parent boundary lines.

Where do I find it?

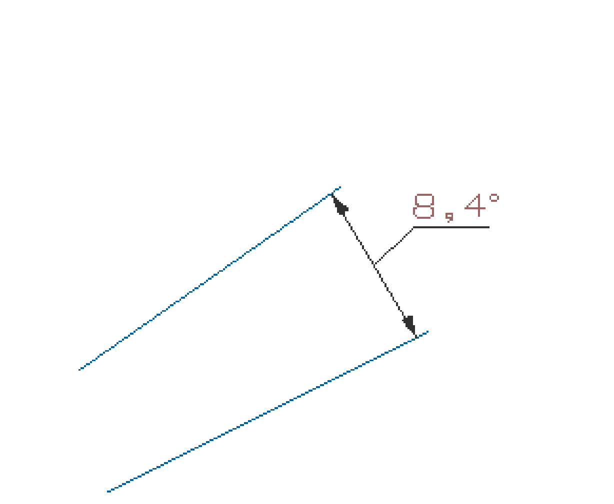
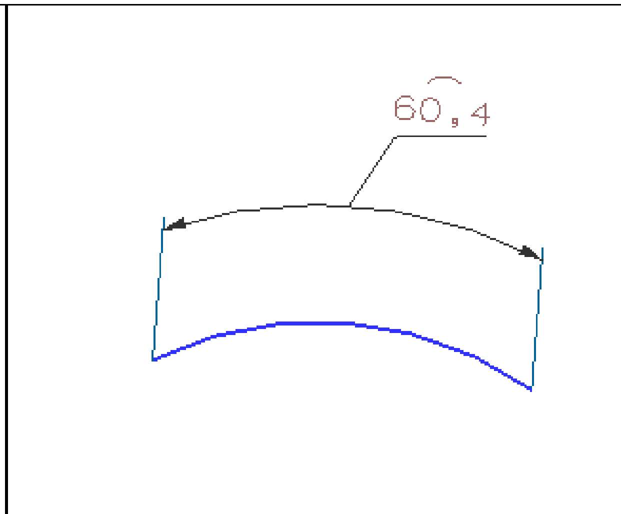
Drafting Standard Customer Default

Application	Drafting
Prerequisite	Drawing views with standard orientations
Menu	**Preferences** or **Style**→**View**→**Detail** tab.
Location in dialog box	**File**→**Utilities**→**Customer Defaults**→**Drafting**→**General**→**Standard** tab →**Drafting Standards**→**View**→**Detail View** tab
Shortcut menu	Graphics window →right-click→**Style** (on a view boundary)

Narrow Arc Length and Angular Dimensions

What is it?

The **Narrow** formatting option shows the value of a small dimension outside the dimension lines. A label shows to which dimension the value applies. Included in the Narrow angle and Narrow arc length dimension options is the ability to change the leader attachment location for all Narrow dimensions (Linear, Arc Length, and Angular).

	
Narrow angle dimension	Narrow arc length dimension

Why should I use it?

Use the new dimensions to make small arc length and angular dimensions easier to read.

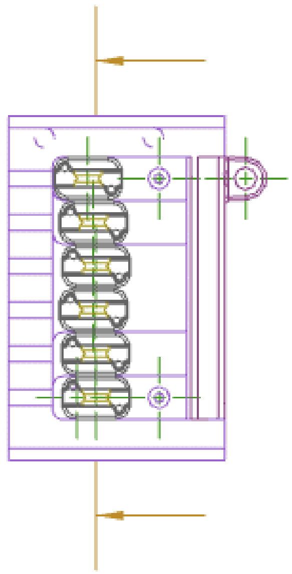
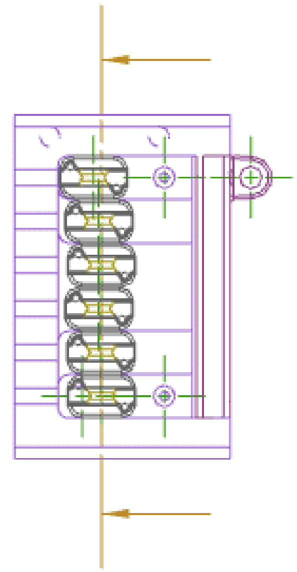
Where do I find it?

Application	Drafting
Prerequisite	Drawing views with standard orientations
Menu	**Preferences**→**Annotation**→**Dimensions** tab→**Narrow**

Section Line Style Enhancements

What is it?

New Section Line Style controls support the ESKD (Russian) and ISO128 Standards. The ESKD Standard includes control for the length of the end line. The ISO128 Standard for section lines has been enhanced to include an Invisible font and control over whether the label is displayed on the arrow or at the end of the section line.

	
ESKD Section Line	ISO128 Section Line

The Invisible font is only available for section line types that support thick ends and breaks. If you set the line font to invisible, the ends and breaks display, but not the lines between them.

Why should I use it?

Use the new Section Line Style controls when support for ESKD or ISO128 is required.

Where do I find it?

Preferences Section Line

Application	Drafting
Prerequisite	Drawing views with standard orientations
Menu	**Preferences→Section Line**
Shortcut menu	Graphics window →right-click→**Style** (on a section line)

Annotation Style Section Line

Application	Drafting
Prerequisite	Drawing views with standard orientations
Menu	**Edit→Annotation Style** and select **Section Line**.
Shortcut menu	Graphics window →right-click→**Style** (on a section line)

PMI

PMI objects in JT files

What is it?

When you open a JT file in NX, the following objects now appear if they were saved in the JT file:

- Assembly-level PMI and model views from Teamcenter JT files. This includes the display of component PMI in assembly model views.
- PMI locator symbols that were created in I-deas or NX.

Why should I use it?

JT files that are opened in NX now more closely match their source files.

Where do I find it?

Toolbar	**Standard**→**Open**  →select a JT file
Menu	**File**→**Open**→select a JT file

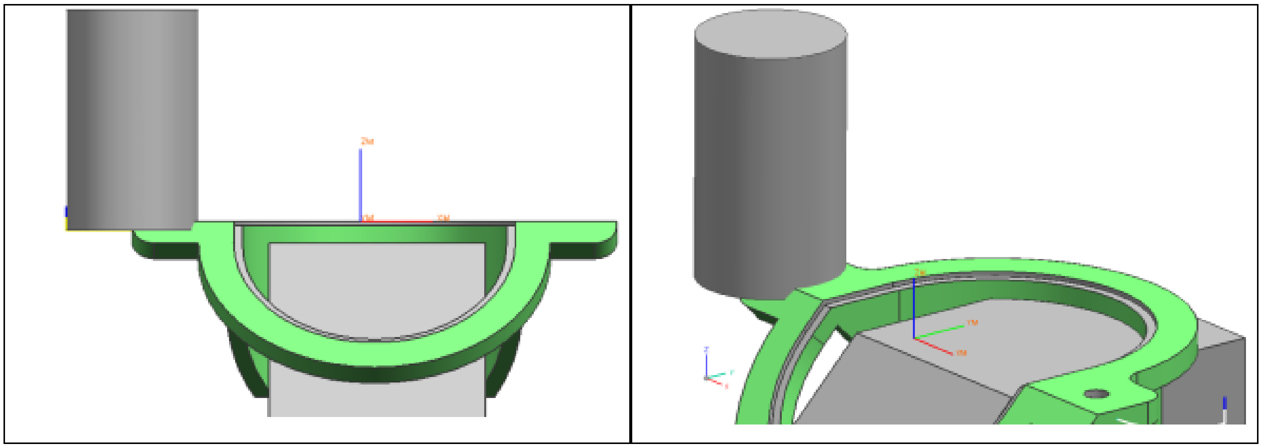
Manufacturing

Face Milling — negative floor stock

What is it?

You can now enter a negative value for **Final Floor Stock** in Face Milling operations.

In Face Milling operations, the cut region is computed assuming 0.0 floor stock. When you define a negative floor stock value, cutting moves created on the faces are moved down into the part by the given value. All moves in the cutting plane are pushed down. This includes cutting, first-cut, and step-over moves.



Why should I use it?

Enter a negative value to make small adjustments to the cut depth in order to meet a specific requirement, such as a particular dimension or tolerance.

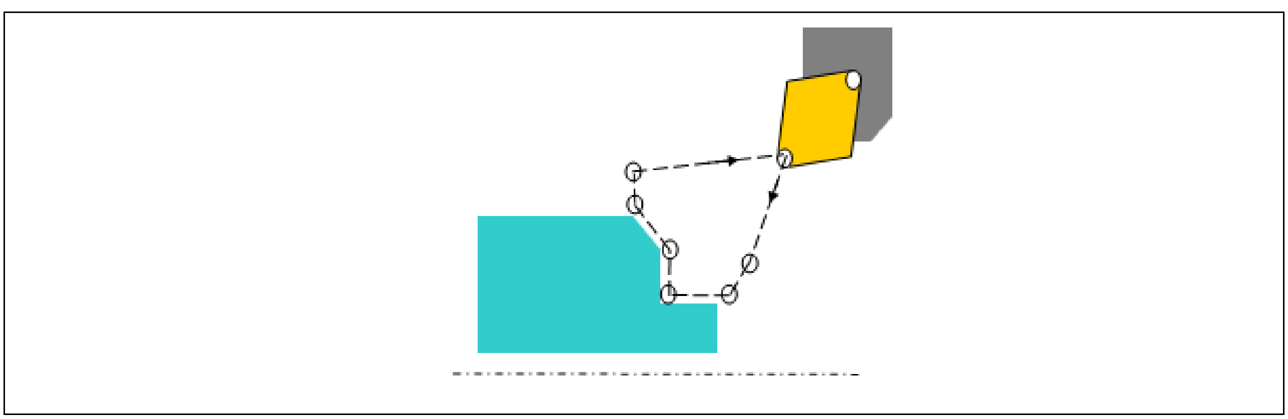
Where do I find it?

Application	Manufacturing
Toolbar	**Insert→Face Milling**
Dialog Box	**Cutting Parameters→Stock** tab→**Final Floor Stock**

Turning cutter compensation

What is it?

Cutter compensation (cutcom) options control the cutter radius compensation function of the machine controller. The software outputs G41 or G42 code to adjust the tool path left or right, measured relative to the tool's direction of motion.



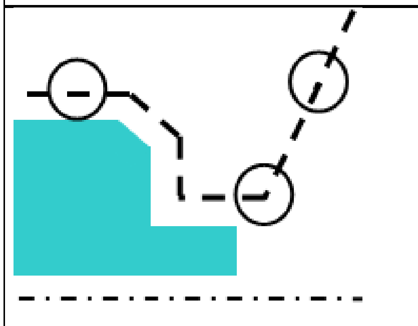
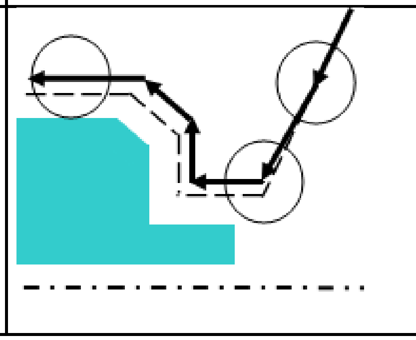
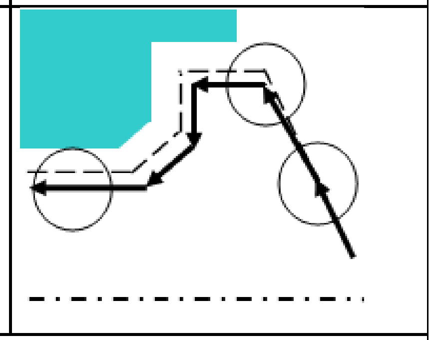
Profile cutting pass without cutter compensation

There are two types of cutter compensation available:

- Fixed tracking point
- Contact contour

Fixed tracking point cutter compensation

If you do not select contact contour cutter compensation, the software generates cutter compensation for tool paths using a fixed tracking point on the tool. This method typically requires an offset value that is small compared to the radius of the tool.

No cutter compensation	Tool path adjusted to the right	Tool path adjusted to the left
		
Programmed tool path Tool path adjusted by controller Tool nose radius		

Contact contour cutter compensation

Contact Contour is also referred to as full radius cutter compensation or material edge contouring. Use contact contour cutter compensation for tool paths generated at the circumference of the tool radius. Contact contour cutter compensation:

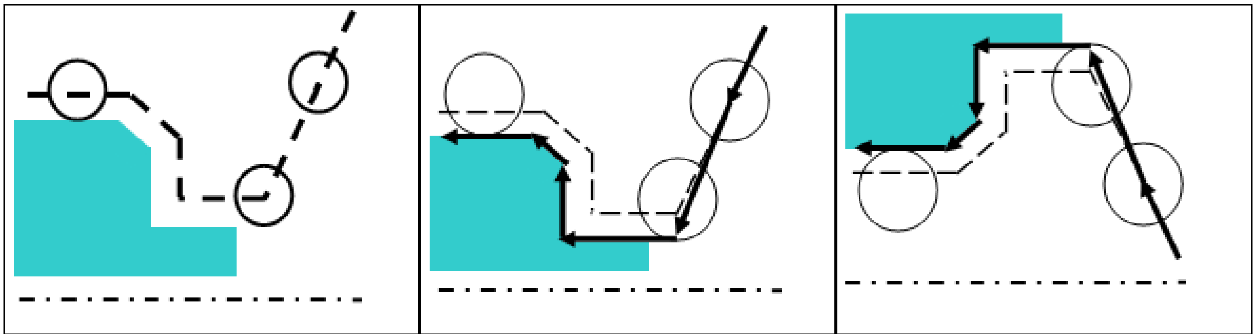
- Is activated with the **Output Contact/Tracking Data** option.
- Ignores any tracking point.
- Measures gauge lengths from the center of the active tool nose radius.

Changing the tool nose radius, such as for a grooving tool, also moves the gauge length measurements to the center of the newly active tool nose radius.

- Typically requires an offset value that is equal to the radius of the tool.

No cutter compensation	Tool path adjusted right to part contour	Tool path adjusted left to part contour

Contact contour cutter compensation



Programmed tool path

Tool path adjusted by controller

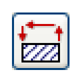
Tool nose radius

Why should I use it?

If the NC programmer turns on cutter compensation, the machine operator can adjust the tool path with the offset value in the cutter compensation register of the controller. This enables the machine operator to:

- More closely control cutting dimensions of the part.
- Compensate for variation in the tool size.
- Compensate for tool or part deflection.

Where do I find it?

Application	Manufacturing
Available for	Turning finishing operation and profiling passes in a Turning roughing operation
Location in dialog box	Operation dialog box→**Non Cutting Moves**  →More page→**Cutter Compensation** group

Divide by Holder

What is it?

Divide by Holder lets you divide a generated tool path where there are holder collisions. When the software finds a holder collision, it moves backwards along the tool path to the closest transfer, then divides the tool path into two operations at the transfer. The original operation contains collision free tool path. The new operation contains the tool path section with holder collisions.

Name	Path	Edit Status
CAVITY_MILL		Repost

Operation Navigator before Divide by Holder

Name	Path	Edit Status
CAVITY_MILL		Repost
CAVITY_MILL_DIV_1		Repost

Operation Navigator after Divide by Holder

You can change the following parameters in the split operation to eliminate holder collisions:

- Tool

  The tool shape, diameter, corner radius, and taper angle must be the same.
- Holder
- Program group
- Machine Control options
- Feed rate

Divide by Holder is useful for:

- Level based operations.
- Area milling operations with a **Zig** or **Zig Zag with Lifts** cut pattern that generates transfers.

  Divide by Holder does not work if the operation does not have transfers. For example, you cannot use **Divide by Holder** for a **Zig Zag** cut pattern where the tool enters the cut, zig-zags everywhere to machine the part, then retracts.

Why should I use it?

You can:

- Generate operations with holder checking turned off for faster performance, then run **Divide by Holder** to remove holder collisions.
- Use **Divide by Holder** when there are changes to the holder information, to correct the tool path without regenerating the operation.

Where do I find it?

Application	Manufacturing
Shortcut menu	**Operation Navigator**→right-click the operation→**Tool Path**→**Divide by Holder**

Where do I find it?

Post Builder template posts

Application	Post Builder V 6.2 and V 7.0
Menu	**File→New**
Location in dialog box	**Controller** group→**Library** list

Sinumerik 840D user defined events

Application	Manufacturing
Menu	**Insert→Operation**
Location in dialog box	Operation dialog box→**Machine Control** group→**Edit** (**Start of Path Events**)

NX Sheet Metal

Tool ID attributes display in the Flat Pattern view

What is it?

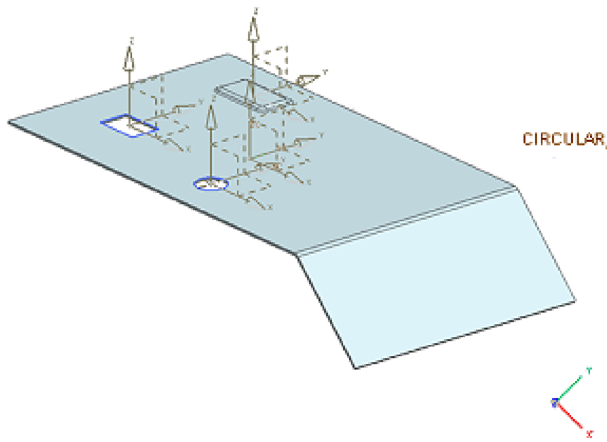
You can display tool IDs as callouts in the Flat Pattern view or drawing member view if the callout text is inserted as a specific attribute (**SHEET_METAL_TOOL_ID**) on a datum CSYS on a Sheet Metal part.

The datum CSYS can be placed anywhere on the Sheet Metal part or it can be associated with user defined features (UDF) or other Sheet Metal features: for example, Dimple, Louver, or Bead.

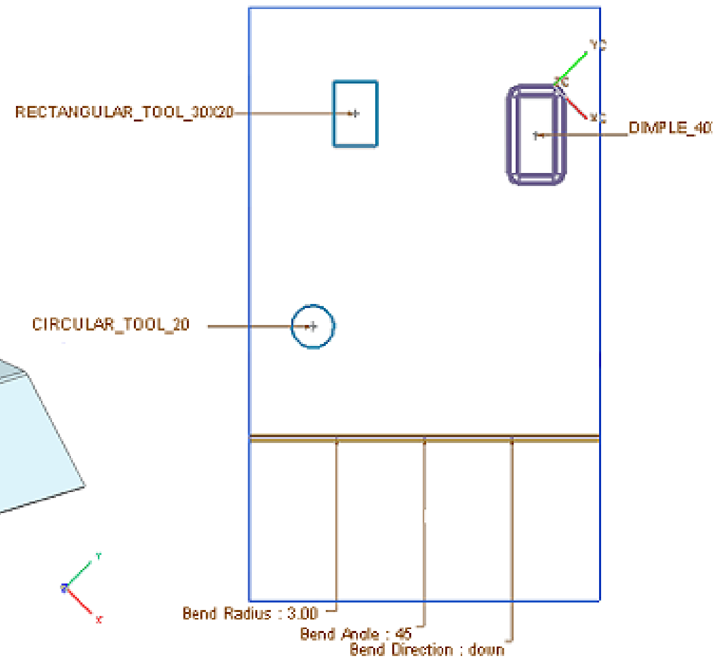
You can also create a tool using UDFs with an associated datum CSYS and insert the attribute, **SHEET_METAL_TOOL_ID**, with the callout text as the attribute value on the datum CSYS. You can then place the tool on the Sheet Metal part using a CSYS or other positioning dimensions. The location of the CSYS on the UDF is also the location of the tool ID callout.

In the following example, two UDFs and a Dimple are placed on the Sheet Metal part. The Rectangular Tool and Dimple are placed using the CSYS for placement, while the Circular Tool is placed using positioning dimensions.

Each tool has the respective callout text **RECTANGULAR_TOOL_30X20**, **CIRCULAR_TOOL_20**, and **DIMPLE 40X20**, inserted as the attribute value for the string attribute **SHEET_METAL_TOOL_ID**. The tool IDs for each tool are displayed in the flat pattern view with a marker, a leader, and the tool ID callout.



Formed Sheet Metal part with UDFs and a Dimple



Tool IDs displayed in the Flat Pattern view

You can modify the tool ID name and control the display of the tool ID using the options in the **NX Sheet Metal Preferences** dialog box. You can also specify customer defaults in Drafting and Sheet Metal for the color, type, width, and layer of the tool ID.

In the Drafting application, you can right-click the tool ID callout, choose **Edit**, and use the **Note** dialog box to edit the callout. You can edit the tool ID Attribute, tool parameters, and location after you place the tool.

Why should I use it?

Displaying tool ID attributes as callouts in the flat pattern view help you identify the tools used to create sheet metal features.

You can use the API in your own NX Open programs to extract information like the tool ID, location, orientation, and tool parameters if they are inserted as attributes. You can extract tags and the bend radius, angle, and neutral factor values associated with flat pattern or flat solid objects. This is helpful in manufacturing processes.

Where do I find it?

Tool ID callouts must be inserted as attribute values of a specific string type of feature attribute, **SHEET_METAL_TOOL_ID**, on a datum CSYS anywhere on a Sheet Metal part, or can be associated with a UDF or other Sheet Metal features.

Flat Pattern Display preferences for tool ID attributes

What is it?

The following NX Sheet Metal preferences are available for flat pattern display:

Tool Marker

Controls the display of the color, font, width, and layer of tool marker objects.

If you clear the **Tool Marker** check box, the **Marker**, **Leader**, and **Tool ID Attribute** values in the flat pattern display are not available.

Tool Id

Displays **Tool ID Attribute** values as callouts in the flat pattern display.

Note

You can set the default values for the tool ID attributes using the customer default options. For more information, see Customer defaults for tool ID attributes.

Where do I find it?

Application	NX Sheet Metal
Menu	**Preferences→NX Sheet Metal**
Location in dialog box	**NX Sheet Metal Preferences** dialog box→**Flat Pattern Display** tab→**Curves** group→**Tool Marker** check box **NX Sheet Metal Preferences** dialog box→**Flat Pattern Display** tab→**New callouts from defaults** group→**Tool Id** check box

Customer defaults for tool ID attributes

What is it?

You can set the default values for tool ID attributes using the following customer default options:

Tool Marker

- **Enabled**

  Select this check box to specify if Tool Marker Objects should be created and displayed on flat patterns.

- **Color**, **Font**, **Width**, **Layer**

  Specifies the color, font, width, and layer for Tool Marker Objects on flat patterns.

Custom Callout 7

- **Available**

  Select this check box to specify if the callout is available on the **Preferences** and **Style** dialog boxes for flat patterns.

- **Enabled**

  Select this check box to specify if the callout is created on flat patterns.

- **Name**, **Object Types**, **Content**

  Specifies the name, object types, and content for the Tool Marker Objects.

Where do I find it?

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Customer Defaults** dialog box→**Sheet Metal→Extras→Curves** tab→**Tool Marker** group **Customer Defaults** dialog box→**Sheet Metal→Extras→Annotations** tab→**Custom Callout 7** group **Customer Defaults** dialog box→**Drafting→Extras→Flat Pattern Curves** tab→**Tool Marker** group **Customer Defaults** dialog box→**Drafting→Extras→Flat Pattern Annotations** tab→**Custom Callout 7** group

Mold and Die Tools

Progressive Die Wizard

Progressive Die Wizard uses assembly constraints

What is it?

Progressive Die Wizard can create assembly constraints when you:

- Add insert groups.
- Add standard parts.

The use of constraints is automatic when you set your assembly positioning preference to **Assembly Constraints**.

Why should I use it?

You should consider using constraints as soon as it is practical because constraints are the current and preferred method of positioning components.

Where do I find it?

If your company already works with assembly constraints instead of the legacy mating conditions you need take no further action.

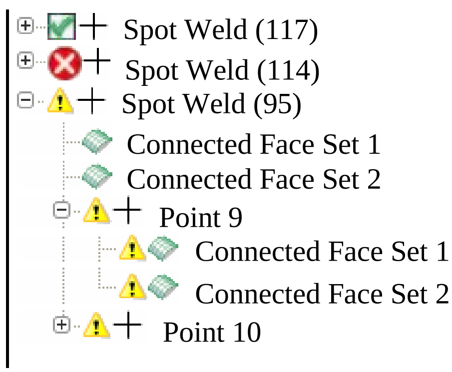
Application	Progressive Die Wizard
Menu	**Preferences → Assemblies → Assembly Positioning Interaction → Assembly Constraints**
Customer Default	**File → Utilities → Customer Defaults → Assemblies → Extras → Assembly Constraints → Positioning → Assembly Constraints**

Weld Assistant

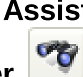
Delete in Connected Face Finder

What is it?

The **Delete** option is now available in the **Connected Face Finder** dialog box. You can delete a weld point directly from the Results tree after you run a Connected Face Finder check.



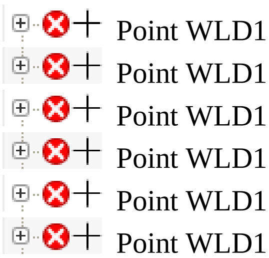
Where do I find it?

Application	Modeling
Prerequisite	Run the **Connected Face Finder**.
Toolbar	**Weld Assistant→Connected Face Finder** 
Menu	**Insert→Welding→Connected Face Finder**
Location in dialog box	In the Results tree, right-click a Point node→**Delete**.

Delete in Weld Advisor

What is it?

The **Delete** option is available in the **Weld Advisor** dialog box. You can delete a weld point directly from the Results tree after you run a Weld Advisor check.



Where do I find it?

Application	Modeling
Prerequisite	Run the **Weld Advisor**.
Toolbar	**Weld Assistant→Weld Advisor** 
Menu	**Insert→Welding→Weld Advisor**
Location in dialog box	In the Results tree, right-click a Point node→**Delete**.

New Auto Point options

What is it?

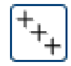
The **Auto Point** command has the following new options.

Maximum Single Thickness Specifies the maximum single metal thickness for all the selected components. If the distance between the top faces of two panels, or sheets, is greater than the specified Maximum Single Thickness plus the specified Maximum Face Gap distance, weld points are not created at that location.

Maximum Bend Radius Specifies the bend radius of a flange. Weld points are not placed on faces with a radius smaller than this value.

Minimum Flange Width Specifies the minimum flange width. If the opposite sides of a flange are smaller than the Minimum Flange Width, weld points are not created on that flange.

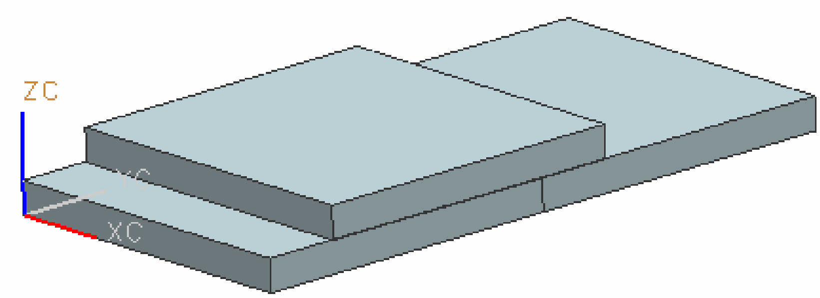
Where do I find it?

Application	Modeling
Toolbar	**Weld Assistant→Auto Point** 
Menu	**Insert→Welding→Auto Point**
Location in dialog box	**Settings** group→**Maximum Single Thickness**/**Maximum Bend Radius**/**Minimum Flange Width**

Reverse Z Direction in Weld Point

What is it?

Reverse Z Direction is a new option in the **Weld Point** dialog box that is available for all spot welds. It lets you reverse the weld CSYS Z vector direction for all welds in the feature set at once, as shown in the following animation.



Why should I use it?

This option is primarily useful when you use sheets instead of solid bodies to represent sheet metal panels. With a sheet, it is not easy to determine the inside and outside of a part, and the Z vector of the weld CSYS may point in an undesirable direction. With the new **Reverse Z Direction** option, you can have better control over the direction of the weld CSYS.

Where do I find it?

Application	Modeling
Prerequisite	Available only when you select the **Display CSYS** check box in the **Preview/Edit** group.
Toolbar	**Weld Assistant→Weld Point**
Menu	**Insert→Welding→Weld Points**
Location in dialog box	**Preview/Edit** group→**Reverse Z Direction**

Control Direction in Weld Point

What is it?

There are now **Control Direction** options in the **Weld Point** dialog box that you can use to control the Z-axis direction for spot welds.

Default	Specifies the direction of the weld CSYS along the Z-axis using the default direction.
Opposite	Specifies the direction of the weld CSYS along the Z-axis using the opposite direction.

Note

The default Z direction may be different for different solid types. For example, a cylinder's default Z direction is opposite of the default Z direction for a measurement vector.

Why should I use it?

You can use **Control Direction** options in conjunction with the **Reverse Z Direction** option for better flexibility in defining connections such as self pierce rivets.

Where do I find it?

Application	Modeling
Prerequisite	Available only when you create Clinch type weld points or custom spot welds that use either a cylinder or cone as the display object.
Toolbar	**Weld Assistant→Weld Point**
Menu	**Insert→Welding→Weld Point**
Location in dialog box	**Settings** group→**Control Direction** list→**Default**/**Opposite**

Ship Design

Manufacturing features

Reference Line

What is it?

Reference Line creates a curve lying in a plane parallel to a grid plane at a user-specified offset.

> **Note**
>
> The grid plan is a datum plane created by the **Concept** module.

Why should I use it?

Use this line as a point to measure from.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Reference Line**
Menu	**Insert→Manufacturing→Reference Line**

Marking Line

What is it?

Marking Line creates curved geometry showing the location of a profile and/or plate on a section of the hull, deck, or bulkhead.

Why should I use it?

In the context of a section assembly, it is necessary to indicate the location of profile and plate parts on a hull section, deck, and/or bulkhead. To accomplish this, you need to create curved geometry with assigned attributes. This geometry and attributes will be part of the output file for the flame cutter. The flame cutter will scribe this information on the part.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Marking Line**
Menu	**Insert→Manufacturing→Marking Line**

Plate Preparation

What is it?

Plate Preparation flattens non-planar plates and applies a shrink factor to all plates.

Why should I use it?

Use **Plate Preparation** to prepare all plates for manufacturing by flattening the plates and/or adding a shrink factor.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Plate Preparation**
Menu	**Insert→Manufacturing→Plate Preparation**

XML Output

What is it?

This function will outputp flame cutter information used for the cutting of parts and scribing information on the parts.

Why should I use it?

Use this functionality to create an XML file, which is used to generate a file that can be read by the flame cutter to manufacture a part.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→XML Output** 
Menu	**Insert→Manufacturing→XML Output**

 Material Allowance

What is it?

Material Allowance creates attributes on the end faces of profiles. The function creates the attribute MK_TYPE and MK_SIZE.

Why should I use it?

This allowance is needed to compensate for manufacturing and assembly inaccuracies.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Material Allowance** 
Menu	**Insert→Manufacturing→Material Allowance**

 Vent Hole Marking Sketch

What is it?

Vent Hole Marking Sketch creates a table showing the location of the ventilation holes on a part.

Why should I use it?

Use this function to create a table which shows the X and Y location of each ventilation hole on a part in the flattened state.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Vent Hole Marking Sketch**
Menu	**Insert→Manufacturing→Vent Hole Marking Sketch**

Knuckled Profile

What is it?

Knuckled Profile creates a bend table for profiles that are bent on a brake press.

Why should I use it?

In order to manufacture bent profiles from a straight semi-finished material, you need to provide a distance measurement from an edge to the bend centerline of each bend. **Knuckled Profile** provides this capability.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Knuckled Profile**
Menu	**Insert→Manufacturing→Knuckled Profile**

Inverse Bending

What is it?

Inverse Bending creates curved geometry on a profile.

Why should I use it?

In order to form curved profiles from a straight semi-finished material, it is necessary to mark a curve on the profiles. This curve is used as a gauge during the bending process. When the curved line becomes straight the profile is in its formed shape.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Inverse Bending**
Menu	**Insert→Manufacturing→Inverse Bending**

Profile List

What is it?

Profile List generates a Bill of Material (BOM) of all the profiles contained in a distributed assembly.

Why should I use it?

Use this to obtain the information needed for the BOM of the profiles contained in a distributed assembly.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Profile List**
Menu	**Insert→Manufacturing→Profile List**

Weld Preparation

What is it?

Weld Preparation adds the needed weld information to a body, by modifying the edge of the body for the weld joint type and adding attributes to the body.

Why should I use it?

Use this function to define the weld joint type and to modify the size and shape of the body so the flame cutter can cut the body to its proper size and shape.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing→Weld Preparation**
Menu	**Insert→Manufacturing→Weld Preparation**

Steel features

Profile / Plate

What is it?

The following features are merged in the new **Profile / Plate** feature:

- **Linear Profile**
- **Non-Linear Profile**
- **Linear Sheet**
- **Non-Linear Sheet**

You can now choose one feature to create a profile or plate. You do not need to know if the placement face is planar or non-planar.

> **Note**
>
> The former **Sheet** feature is now called **Plate**.

Why should I use it?

This merger allows you to edit the feature and switch the type from a profile to a plate or from a plate to a profile.

Where do I find it?

Application	Ship Design
Toolbar	**Steel Features→Profile / Plate** 
Menu	**Insert→Steel Features→Profile / Plate**

Endcut

What is it?

This option defines an endcut by utilizing the **Steel Feature Library** function. It also allows you to apply a miter to an endcut feature or to the thickness face of a solid body.

Why should I use it?

Use this functionality to modify the end condition of any solid body.

Where do I find it?

Application	Ship Design
Toolbar	**Steel Features→Endcut** 
Menu	**Insert→Steel Features→Endcut**

 Update Steel Library

What is it?

The reading in of a library part is now a separate step and includes endcuts.

Why should I use it?

This allows you to read in all of your different library parts before you create a feature and updates the library in a part file if the library changes.

Where do I find it?

Application	Ship Design
Toolbar	**Steel Features→Update Steel Library** 
Menu	**Insert→Steel Features→Update Steel Library**

Teamcenter Integration for NX

Password option for utilities when running Teamcenter 8

What is it?

You can hide the password in a file that contains the password. When you run Teamcenter 8, you can use the new password option **pf** as a command line option instead of **p**.

You can use this whenever a password is required. This includes all Teamcenter Integration utilities and command line startup of Teamcenter Integration.

The value for **pf** is a file name. Teamcenter Integration reads the password in the file and sends the extracted password to Teamcenter.

For example, you can use the following to start Teamcenter Integration:

```
ugraf —pim=yes —u=xyz —pf=mypassword.dat
```

It is recommended that you ensure that the file is protected by local machine access controls so that only authorized user accounts for certain applications can read the file.

Why should I use it?

You can keep passwords more secure by hiding them from unauthorized users.

Where do I find it?

Application	Teamcenter Integration command line utilities and command line startup

Adding item types to File New templates

What is it?

You can select from multiple item types for a template when you create a new item. The item types available can vary for each template and are provided in the **Item Type** list. The listed item types can be customized for each template.

The **Item Type** list can be modified to list several or all valid item types by modifying the template PAX files.

> **Note**
>
> Acceptable item types are those that are valid Teamcenter item types.

The following XML element needs to be modified for each template to add additional item types:

```
<ItemType></ItemType>
```

The following examples show how to modify the XML element for different scenarios:

- To display the full list of item types:

  ```
  <ItemType>Any</ItemType>
  ```

- To display the full list of item types, but have an item type preselected and displayed as the default:

  ```
  <ItemType>NXPart,Any</ItemType>
  ```

- To display a specific list of item types, include only those that are to be listed:

  ```
  <ItemType>NXPart,NXDrawing,NXSheetmetal</ItemType>
  ```

- To keep the template tied to one item type, include only that one:

  ```
  <ItemType>NXPart</ItemType>
  ```

  > **Note**
  >
  > If only one item type is allowed for a template, the **Item Type** list is not displayed in the **File New** dialog box.

The **Item Type** list is only shown when you are creating a new item, not when you are creating a non-master such as a drawing that has a **specification** relation type.

When auto-assign is off, after you assign a number, revision, or item name and the item type is changed, all of the fields are cleared. If auto-assign is on and the item type is changed, all of the fields are reassigned new numbers and values. This occurs because if you have a naming rule in effect for one item type but not another, the values that are current may not be valid.

For CAM templates (in the **Manufacturing** tab in the **File New** dialog box), the list of valid item types is defined by the Teamcenter preference **NX_supported_operation_types**. The functionality for item types in CAM templates is the same as other templates, but limited by the types set in the preference. The item type default for CAM templates is MENCMachining.

Why should I use it?

You can customize new item templates by adding a list of specific item types that you can select.

Where do I find it?

Application	Teamcenter Integration
Menu	**File→New**
Location in dialog box	**Item Type**

Create an alternate representation in model template

What is it?

You can use **File→New** to create a new model as an alternate representation (Alt Rep). On the **Model** tab, select the **Alt Rep** check box for a model template to create an Alt Rep. The Alt Rep dataset is created under an existing Item Revision.

The value for the Alt Rep dataset name is created using the same method that is used for creating an Alt Rep when you perform a **Save As** operation (<ItemNumber>**-**<ItemRev>-alt#).

You cannot create a new item when an Alt Rep is being created. When the Alt Rep check box is selected (it is not selected by default), the **Item Type** list is not available. Also, you cannot create an Alt Rep for CAE and CAM items.

In the modeling PAX file (**nxdm_ugs_model_templates.pax**), the **Alt Rep** checkbox is not included by default and must be added for modeling templates. Modeling templates are defined as only those templates where the **class** attribute of **ObjectData** has the value **ModelTemplate** as in `<ObjectData class=”ModelTemplate”>`. To add the **Alt Rep** check box to a model template, add the following to the PAX file:

```
<ShowAltRep>Yes</ShowAltRep>
```

For example:

```
<PaletteEntry id="d2">
    <References/>
    <Presentation name="Model" desription="NX Example with datum CSYS">
        <PreviewImage type="UGPart" location="@DB/model-plain-1-mm-
template/A"/>
    </Presentation>
    <ObjectData class="ModelTemplate">
        <Filename>@DB/model_inch/A</Filename>
        <Units>English</Units>
        <ItemType>Item,Part,SpecElement,DMTemplate</ItemType>
        <RelationType>master</RelationType>
        <ShowAltRep>Yes</ShowAltRep>
    <ObjectData>
</PaletteEntry>
```

When you have a part loaded and select the **Alt Rep** check box, the template switches to non-master functionality. The **Number**, **Revision**, and **Item Name** boxes are grayed out, the Dataset Name box is displayed, and the **Reference Part** boxes are populated with values. You can use the **Browse** button to select a different reference part. If a part is not loaded, those boxes are blank and grayed out. You can use the **Browse** button to select a part to reference.

In addition, the **Add master part as a component** check box is displayed. If you select this check box, the master dataset is added as a component of the newly created Alt Rep. This is shown in the Assembly Navigator.

If you create an Alt Rep and the referenced item does not have a master dataset or a master dataset exists without a named reference, when you click **OK** in the **File New** dialog box to create the Alt Rep, you are prompted to create the master first.

If you select an existing Alt Rep as the reference part, the corresponding master part is loaded and used as the reference part instead.

A Blank Model template contains the **Alt Rep** check box by default. A Blank Model template is displayed if all of the templates in a PAX file are modeling templates. If the templates in a PAX file are a mixture of modeling and other templates, then the blank template is a Blank Gateway template, and the **Alt Rep** check box is not included.

If you create an Alt Rep in Teamcenter and open it in NX, the model templates that have the **Alt Rep** specification in the PAX file are shown in the **File New** dialog box. If no templates have the **Alt Rep** specification in the PAX file, then all model templates are shown with the **Alt Rep** check box displayed and the **Add master part as a component** check box not displayed. You can create the Alt Rep but no master dataset can be added as a component.

Why should I use it?

You can create an Alt Rep from a new model template.

Where do I find it?

Application	Teamcenter Integration
Menu	**File→New**
Location in dialog box	**Model** tab→**Alt Rep** check box

Storing template PAX files in Teamcenter

What is it?

You can store the PAX files used to define the templates in Teamcenter Items and Item Revisions. The PAX files stored in Teamcenter are read by Teamcenter Integration during startup to create the templates contained in the **File New** dialog box.

You define the location of the PAX files with the following Teamcenter preference:

TC_NX_FileNewPAXFiles_NX?

where ? is the version of NX you are running, such as 6 or 7

For example, if you are running NX 7: **TC_NX_FileNewPAXFiles_NX7**

Valid values: Items or Item Revisions

For example: **model_1** or **model_2/A**

Note

Typically there are multiple values, such as one for the model PAX file, one for the drawing PAX file, and so on.

This preference is not shipped with Teamcenter; you must add it manually. If the preference is not used, Teamcenter Integration defaults to the default location of PAX files used in previous releases of NX, which is **%UGII_BASE_DIR%\ugii\templates**, or as defined by the environment variable **UGII_TEMPLATE_DIR**.

If only an Item is listed in the preference, the PAX file in the latest Item Revision of that Item is used. This is determined by the **Latest by Creation Date** revision rule.

To modify the contents of a PAX file, you revise the Item Revision and then change the PAX file dataset under the newly created Item Revision.

It is recommended that you store each PAX file dataset in a separate Item. This method enables each Item Revision of the Item to contain the same PAX file with different revisions of the file. Storing multiple PAX files in the same Item could cause problems as subsequent Item Revisions could contain different PAX files, not just different versions of the same file.

Why should I use it?

Managing the PAX files in Teamcenter Items and Item Revisions provides more flexibility and universal access to the same PAX files within an installation and allows you to replicate the PAX files in a Teamcenter multi-site configuration.

Where do I find it?

Application	Teamcenter Integration
Menu	**File→New**
Location in dialog box	Templates in tabs

SSL supported in four-tier Teamcenter environment

What is it?

Secure Socket Layer (SSL) is now available when you connect to Teamcenter in a four-tier Teamcenter environment. SSL is a layer of security provided by web servers and its use is identified by a URL that begins with **https**.

SSL capability is provided by the use of Teamcenter's Service Oriented Architecture (SOA) as the new communications channel with Teamcenter in the four-tier environment. SOA replaces AIWS as the communications channel.

Why should I use it?

You have a more secure communication method when using TCIN in a four-tier Teamcenter environment.

Where do I find it?

Application	Teamcenter Integration The incorporation of Teamcenter's SOA is part of the TCIN architecture for this release and is automatically implemented in the software. No preferences or options are needed to activate the functionality.

Performance improvement for four-tier deployment

What is it?

NX four-tier client performance improvements enable NX users from remote divisions and disparate locations to work in real-time with regional/global Teamcenter hubs at network latencies as high as 250ms with four-tier deployments.

With these improvements, you can significantly reduce your IT costs by consolidating your multi-site environments.

> **Note**
>
> This capability is available from NX 6.0.3.

Why should I use it?

You can work globally more easily and efficiently, and reduce IT costs.

Where do I find it?

Application	Teamcenter Integration in a four-tier Teamcenter environment.

Systems Design

PCB Exchange

New drilled holes design features support

What is it?

Previously, you could specify **Drilled Hole Attributes** only for Simple Hole features you created using the **Hole** command in Modeling. Now you can specify **Drilled Hole Attributes** also for cylindrical holes created using the following Modeling commands:

- **Extrude**. The hole must be created by extruding a circular curve using the **Subtract** Boolean option.
- **Cylinder**. The hole must be created by subtracting the cylinder from the board.
- **Instance Feature**. The rectangular or circular array of holes is created by copying a previously created hole using **Hole**, **Extrude**, or **Cylinder** commands.

> **Note**
>
> If you extrude a curve that is not circular, the Extrude feature is not a drilled hole, but only a cutout.
>
> When importing ECAD model, drilled holes are still created as Simple Hole features.

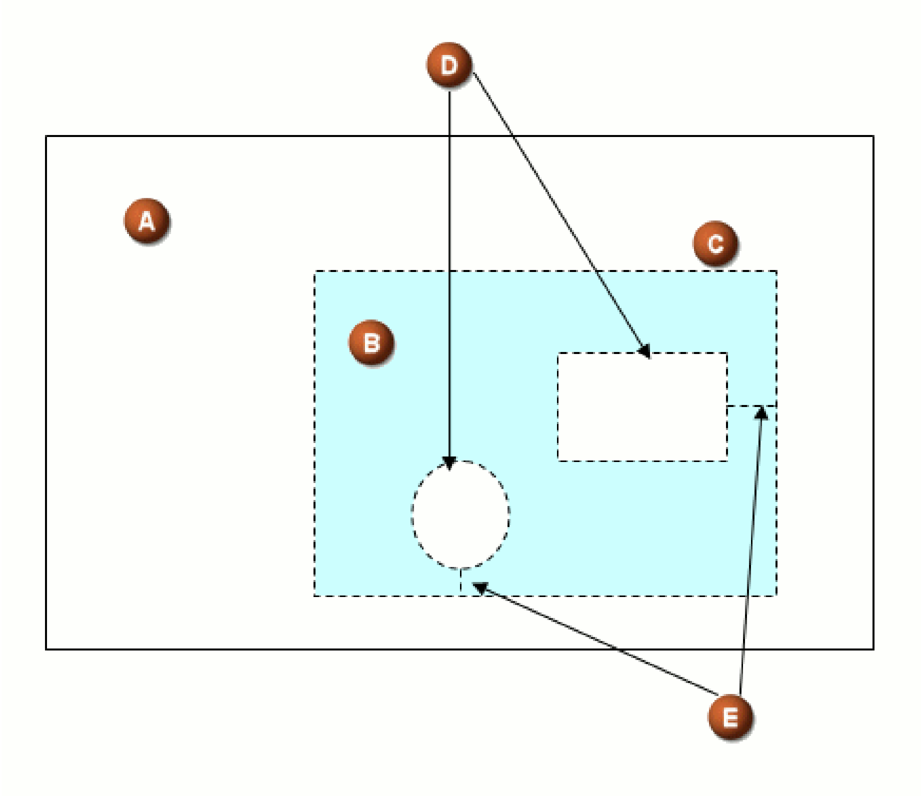
Where do I find it?

Application	PCB Exchange
Prerequisite	An imported ECAD model or an NX CAD printed circuit assembly with holes created in the Modeling application using the **Extrude**, **Cylinder**, or **Instance Feature** commands.
Toolbar	**PCB Exchange** toolbar → **Drilled Hole Attributes** 

Merge restricted areas with cutouts into one loop

What is it?

You can now merge cutouts of a keep-in, keep-out, or other area region with its outer loop to form one loop. PCB Exchange merges loops of restriction areas by introducing a small gap that connects cutouts (inner loops) to the outer loop.



A	PCA board
B	Restricted area (keep-in, keep-out, or other area) in blue
C	Outer loop
D	2 cutouts (inner loops)
E	Gaps between the inner and outer loops

To activate this feature the following variables should be set in *pcbx_ug_model.ini* file:

- `NxWriteMergeAreaLoops = Yes`
- `NxWriteMergeAreaGapSize = #` where `#` is the gap size. By default, this value is 1e-3.

Why should I use it?

Use this option when your ECAD system does not accept the definition of separate restriction area and cutout loops.

Where do I find it?

The *pcbx_ug_model.ini* file can be found in the:

- Network Location specified by MAYA_PCB_ENV_DIR
- Working directory
- Location specified by MAYA_PCB_DIR
- NX installation directory

Display the board coordinate system

What is it?

The board coordinate system is now automatically displayed when the **Board Attributes** dialog box is open.

The board coordinate system is only visible when you use the **Board Attributes** command.

Where do I find it?

Application	PCB Exchange
Prerequisite	An imported ECAD model or an NX CAD printed circuit assembly
Toolbar	**PCB Exchange** toolbar → **Board Attributes** 

Override board thickness

What is it?

You can now override the board thickness from the ECAD model by selecting the **Override Board Thickness** check box and entering a value in the **Board Thickness** box in the following dialog boxes:

- **Import ECAD Model**
- **Compare and Update PCA**

You can change the default value in the **Board Thickness** box by modifying the `NxWriteDefaultBoardThk` variable in the *pcbx_ug_model.ini* file.

Why should I use it?

Use this option when the board thickness defined in the ECAD file is not correct.

Where do I find it?

Application	PCB Exchange
Toolbar	**PCB Exchange** toolbar → **Import ECAD Model** / **Compare and Update PCA**

Specify drilled hole associated part

What is it?

The **Associated Part** option in the **Drilled Hole Attributes** dialog box is now a list with Intermediate Data Format (IDF) accepted choices. In previous releases, this option was a box in which you entered the name of the associated part.

You can now choose the associated part on the board from the **Associated Part** list. The choices on the **Associated Part** list are:

- **BOARD** — The hole is associated to the board part.
- **NOREFDES** — The hole is associated to a mechanical component.
- **Specify** — The hole is associated to an electrical component. You must specify the name of the component in the **Specify Part** box.

Where do I find it?

Application	PCB Exchange
Prerequisite	An imported ECAD model or an NX CAD printed circuit assembly with drilled holes
Toolbar	**PCB Exchange** toolbar → **Drilled Hole Attributes**

Default directory for ECAD files

What is it?

You can now specify a default directory for ECAD files.

You can enter the absolute path to the directory in one of the following:

- The **Default ECAD Directory** box in the **Settings** dialog box.
- The `EcadDefaultDir` variable in the *pcbx_ug.ini* file.

Note

The PCB Exchange browser always points to the specified default ECAD directory. If you do not specify the default ECAD directory, the PCB Exchange browser points to the last selected directory.

Where do I find it?

Application	PCB Exchange
Menu	**PCB Exchange → Settings**
Location in dialog box	**Default ECAD Directory**

Directory for new created components

What is it?

The **Create New Components In** option is now available in the **Settings** dialog box. It is no longer available in the **Import ECAD Model** and **Compare and Update PCA** dialog boxes.

PCB Exchange places files containing new components from the ECAD footprint in the directory you specify in the **Create New Components In** box.

In the **Create New Components In** box, you can enter the absolute path to the directory, the Teamcenter folder name, or the following reserved keywords:

- `ECAD_FILE_DIR` — Use this keyword to place new components in the same directory as the ECAD file.
- `NX_PART_DIR` — Use this keyword to place new components in the same directory as the NX assembly or model file.

You can also set the directory for new components in the `NewComponentDir` variable of the *pcbx_ug.ini* file.

Where do I find it?

Application	PCB Exchange
Menu	**PCB Exchange → Settings**
Location in dialog box	**Create New Components In**

PCA import rules

What is it?

You can now setup rules on how NX represents a printed circuit model. Prior to this version, the board was always defined in the master geometry of the printed circuit assembly (PCA) part. With these new design rules, you can configure PCB Exchange so that the board is imported in a separate part and added as an assembly component to the main PCA part.

On the PCA page of the **PCB Exchange Settings** dialog box, you can specify how to:

- Name the PCA. You can:
  - Use the current NX model name.
  - Use the ECAD model name.
  - Specify a PCA name at import.
- Group PCA entities. This option creates subassemblies in order to classify the printed circuit entities.
- Import the PCA board, components, and areas. You can import each entity as a master geometry of the main PCA part or an assembly component.

Why should I use it?

PCA import rules give you more flexibility when importing PCA entities into NX.

Where do I find it?

Application	PCB Exchange
Menu	**PCB Exchange → Settings**
Location in dialog box	PCA page

PCB Exchange Settings dialog box enhancements

What is it?

The **PCB Exchange Settings** dialog box now has the following tabs:

- **General**
- **PCA**
- **Board**
- **Holes**
- **Components**
- **Keep-ins**
- **Keep-outs**
- **Other Areas**
- **Other Entities**

The settings on these different pages let you specify the most common PCB Exchange work environment variables found in *pcbx_ug.ini* and *pcbx_ug_model.ini* initialization files.

Why should I use it?

Instead of modifying the initialization options in the *pcbx_ug.ini* and *pcbx_ug_model.ini* files and then restarting NX in order to have these options take effect, you can now set the most common PCB Exchange settings in the improved **PCB Exchange Settings** dialog box.

Where do I find it?

Application	PCB Exchange
Menu	**PCB Exchange → Settings**

Import ECAD model in a part file

What is it?

You no longer need to import ECAD models into an NX part file in the No Part state. For example, if you have a part file with an electronic enclosure, PCB Exchange allows you to directly import a PCA model into this enclosure's part file.

If the part file contains a valid PCB, in order to avoid confusion, it is strongly recommended that you design this PCB as a subassembly. To share it with the imported ECAD model, just make the PCB the work part or the displayed part.

Where do I find it?

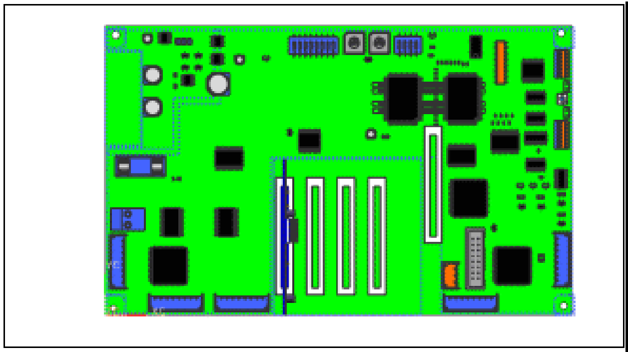
Application	PCB Exchange
Prerequisite	A non empty part file
Toolbar	**PCB Exchange** toolbar → **Import ECAD Model**
Menu	**PCB Exchange → Import ECAD Model**

ECAD/NX model comparison enhancement

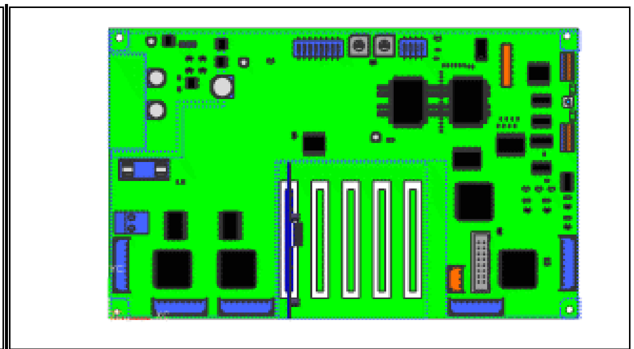
What is it?

When comparing the current NX model with an existing ECAD model using the **Compare and Update PCA** command, you can now preview the detected changes in the graphics window by selecting **Preview Changes**.

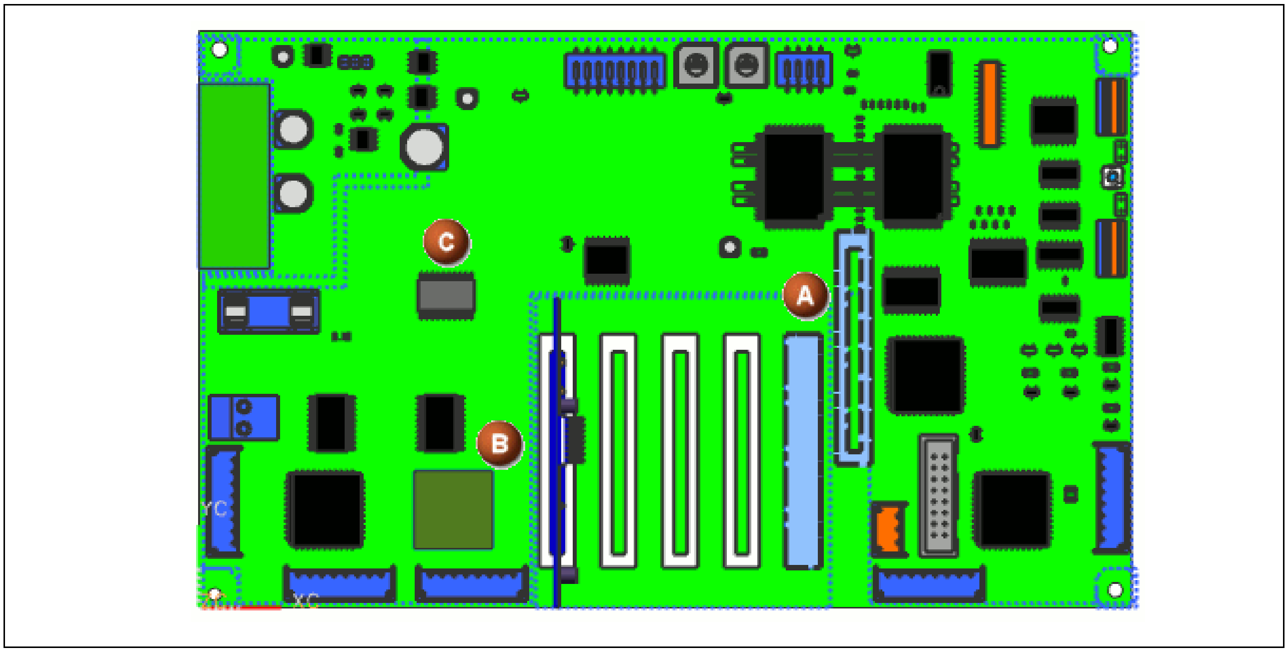
The following graphic shows the original models and a preview of the detected changes.



Original NX model



Original ECAD model



Detected changes between NX and ECAD models

The following changes are detected:

- Component  is not placed at the same location in the NX model and the ECAD model.
- Component  is deleted from the NX model.
- Component  is added to the NX model.

If you select one of the changed components from the component list under the **Components Status** group, the NX model component and the ECAD model component are highlighted in the graphics window.

Why should I use it?

The **Preview Changes**  icon helps you visualize the differences between the NX model and the ECAD model.

Where do I find it?

Application	PCB Exchange
Toolbar	**PCB Exchange** toolbar → **Compare and Update PCA**
Menu	**PCB Exchange** → **Compare and Update PCA**

Filtering enhancements

What is it?

The improved **Filters List** dialog box contains pre-defined and user-defined filter rules.

When you import, export, or compare and update ECAD models, you can now select the entities to remove in the **Filters List** dialog box. You can select:

- Different types of holes.
- Different types of components.
- Different types of restriction areas.

Why should I use it?

The filter rules are now much easier to use with the improved **Filters List** dialog box.

Where do I find it?

Application	PCB Exchange
Toolbar	**PCB Exchange** toolbar → **Import ECAD Model** / **Export ECAD Model** / **Compare and Update PCA**
Menu	**PCB Exchange** → **Import ECAD Model** / **Export ECAD Model** / **Compare and Update PCA**
Location in dialog box	In **Import Options**, **Export Options**, or **Compare Options** group, select **Use Entity Filter** / **Use Entity Filter on NX model** / **Use Entity Filter on ECAD model** → **Filter List**

PCB Exchange for Zuken

What is it?

PCB Exchange for Zuken is a new application that lets you import or export PCB assemblies in the Zuken's CR5000 native format.

You import or export Zuken format files the same way you import or export files of other supported formats.

All functionality previously supported for other file formats is also supported for the Zuken format. In addition, the Zuken format supports the import and export of:

- Flexible printed circuits
- Traces
- Pads
- Metal masks
- Resist masks

On the Other Entities page of the **PCB Exchange Settings** dialog box, you can specify how to:

- Import traces, pads, masks, and internal layers from Zuken format files in the **Import Rules** group.
- Export projection view of the PCB to Zuken format files in the **Export Rules** group.

By default, PCB Exchange for Zuken imports and exports to version 10 of the CR5000 application. You can change this to version 9 by setting the following environment variable `UGII_PCB_CR5000_VERSION` to 9.

Where do I find it?

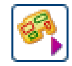
Note

When importing PCB assembly from Zuken format files, file type is automatically detected.

To specify Zuken import and export settings

Application	PCB Exchange for Zuken
Prerequisite	An active PCB assembly
Menu	**PCB Exchange → Settings**
Location in dialog box	Other Entities page

Export PCB assembly to Zuken format files

Application	PCB Exchange for Zuken
Prerequisite	An active PCB assembly
Toolbar	**PCB Exchange** toolbar → **Export ECAD Model** 
Menu	**PCB Exchange** → **Export ECAD Model**
Location in dialog box	**Export Options** → **File Format** → **Zuken bmb** or **Zuken pcb**