What's New in NX 6

Proprietary & restricted rights notice

This software and related documentation are proprietary to Siemens Product Lifecycle Management Software Inc.

© 2008 Siemens Product Lifecycle Management Software Inc. All Rights Reserved.

All trademarks belong to their respective holders.

Contents

Introduction to What's New 1-1

Synchronous Modeling with synchronous technology 2-1

User efficiency 3-1

Designing in an assembly context 4-1

Teamcenter Integration for NX 5-1

Non-geometric components in NX assemblies 5-1
Parts list 5-2
Additional columns in Assembly Navigator 5-3
Refresh Teamcenter Information 5-4
Reset columns 5-5
Quantity in an assembly 5-5
Reopen parts modified by other users 5-6
Assign items to projects 5-7
Teamcenter columns in NX 5-8
Freeze Column 5-9
Projects in Teamcenter Navigator 5-10
Item name 5-10
Import assembly 5-11
Teamcenter Integration commands moved 5-11
Import part with validation options 5-12
Template setup 5-14

NX Essentials 6-1

Full screen display 6-1
True Shading 6-3
Selection MiniBar 6-4
Customizable radial toolbars 6-5
Resource Bar toolbar 6-7
Toolbar Manager 6-8
Shortcuts 6-9
De-emphasis settings 6-10
Vector Constructor 6-11
CSYS Constructor 6-13
OrientXpress tool 6-13
Allow multi-select of hidden faces 6-14

Preferences 6-15
Color dialog box 6-15
Wireframe Contrast 6-16
Grid and Work Plane 6-17
Automatic model size preference for facets 6-18
Datum plane grid 6-19
Exit and Close 6-20
Hide objects immediately 6-21
View Section 6-21
Move Object 6-22
Decal 6-23
Layer Settings 6-24
Movie capture in NX 6-25
Text export in CGM and PDF files 6-26

Design 7-1

Modeling 7-1
Synchronous Modeling with synchronous technology 7-1
Hole 7-15
Blend enhancements 7-17
Boolean enhancements 7-18
Split Body 7-19
Replace Feature enhancement 7-20
Mirror a CSYS using Instance Geometry 7-20
Active Selection 7-22
Feature Replay 7-23
Planes and datum planes using the view plane 7-24
Planes and datum planes using On Curve 7-24
Patch Openings 7-25
Extended basic expression data types 7-26
Selection Intent in Extract 7-27
Global Shaping by Function 7-28
Point Set 7-29
Copying faces 7-29
Datum CSYS 7-30
Block, Cylinder, Sphere, and Cone associativity 7-31
Assemblies 7-31
NX Relations Browser 7-31
Assembly constraint filtering 7-32
Assembly Navigator constraint filtering 7-33
Geometry selection for designing in an assembly context 7-33
Facet selection 7-37
Deform Components 7-37
WAVE General Relinker 7-38
Replacement Assistant 7-39
Move Component 7-40

Replace Component 7-41
Selection intent in WAVE Geometry Linker 7-41
Assembly Navigator 7-42
Restore last NX session 7-42
Reference sets 7-43
Load controls for multi-CAD JT files 7-44
Sketcher 7-45
Trim Recipe Curve 7-45
Orient Sketch on Path axes to a face or a curve 7-46
Sketch Style, Parameters, and Preferences Changes 7-47
Shape Studio 7-48
Best Fit Alignment 7-48
Fill Hole 7-49
Rapid Surfacing 7-50
Deviation Gauge 7-51
Drafting 7-52
View dialog box enhancements 7-52
Annotation Leader and Origin dialog box enhancements 7-52
Leader on screen interaction 7-53
Sketch in Drafting 7-54
2D and 3D Centerlines 7-54
Centerline handles 7-55
Store custom symbol in part 7-56
Smash Custom Symbol 7-56
Feature Control Frame 7-57
New datum target types 7-58
Crosshatch 7-59
Wireframe color from face 7-59
Oriented Section View 7-61
Offset section line 7-62
Sectioned/Non-sectioned solid bodies 7-64
Datum terminator 7-64
Infer diameter dimension for full circles 7-65
Interference curves 7-65
Associativity for ordinate dimensions 7-66
Import Drafting Standard 7-67
PMI 7-67
Regions 7-67
Bidirectional edits of PMI 7-69
Sheet Metal 7-70
NX Sheet Metal 7-70

Data Reuse 8-1

Reuse Library 8-1
Feature/Object template 8-1
Create a Feature/Object template 8-1

Filter Member Select view 8-3
Reuse Library-Fastener Assembly 8-3
Shape Search-Geolus integration 8-4

Systems Design 9-1

Routing Systems 9-1
Routing Systems: General 9-1
Routing Systems: Electrical 9-6
Automotive Applications 9-10
General Packaging 9-10
Die Design 9-17
Die Engineering 9-22
Die Validation 9-23
Ship Design 9-30
Steel Features 9-30
Flexible Printed Circuit Design 9-31
Bridge Transition 9-31
PCB.xchange 9-33
Customizing work environment with site-specific settings 9-33
Commands available on the PCB.xchange toolbar and menu . . . 9-33
Board mesh simplification techniques 9-34
Junction-to-board coupling 9-35
Direct access to NX Electronic Systems Cooling from PCB.xchange 9-35
New options to view restriction areas 9-37

Digital Simulation 10-1

Design Simulation 10-1
3D Tetrahedral Mesh enhancements 10-1
New customer default for resetting model cleanup operations . . . 10-2
Checking node proximity to underlying geometry 10-3
Create and Edit Solution usability improvements 10-4
Materials enhancements 10-4
Advanced Simulation 10-5
Assembly FEM 10-5
Data management 10-7
General capabilities 10-9
Geometry idealization 10-15
Material and physical properties 10-16
Meshing 10-20
Geometry abstraction 10-33
Boundary conditions 10-37
Validating the model 10-40
Solvers and solutions 10-42

NX Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal 10-74
Post-processing 10-102
Composite Laminates 10-106
Response Simulation 10-113
Motion Simulation 10-123
Mechatronics and control 10-123
General functionality 10-126
Usability improvements 10-130
Functions and Graphing 10-134
Persistent graph display settings 10-134
Synchronize functions 10-134
New function types 10-136

Optimization and Analysis 11-1

NX Analysis 11-1
One-step Formability Analysis 11-1
Optimization and Sensitivity Study 11-2
Optimization 11-2
Sensitivity Study 11-4

Product Validation 12-1

Check Requirements 12-1
Save check result to Teamcenter 12-1
Check-Mate 12-1
Check-Mate Treat Warning as Pass 12-1
Checker and function enhancements 12-2
Sheet Metal Validation Checkers 12-2

Manufacturing Design 13-1

Mold and Die Tools 13-1
Mold Design 13-1
NX Electrode Design 13-23
Progressive Die Design 13-25
Weld Assistant 13-37
BIW Locator 13-37
Weld Filter 13-40
Weld Point 13-43
Auto Point 13-43
Export Welds 13-44
Group ID colors 13-45

Manufacturing 14-1

General 14-1
Manufacturing setups available with File® New 14-1

Operation Navigator 14-2
Template changes 14-4
Corner Control 14-5
Cut Patterns 14-5
CAM roles 14-6
IPW color plots 14-6
CAM Geometry toolbar 14-7
Command Finder for Manufacturing 14-8
Journaling and API 14-8
CAM Express 14-9
CAM Express home page 14-9
CAM Express tutorials 14-9
Tools 14-10
Tool display 14-10
Pockets 14-11
Turning tools 14-12
Solid tools for Turning 14-13
Milling and Drilling tools available in Turning 14-14
Sample user defined milling tools in the tool library 14-15
Milling 14-16
Tool path smoothing 14-16
Face Milling 14-17
2D/3D Profiling 14-19
Thread Milling 14-21
Fixed and variable surface contouring 14-22
Plunge Milling 14-27
Finish passes and cutter compensation 14-28
Zlevel 14-29
Turning 14-30
New look for Turning 14-30
Multi-function machines 14-30
Approach and Departure 14-31
Custom boundary offsets 14-31
Finish operation cut order 14-32
IPW 14-33
Local returns and cycle interrupts 14-35
Feed rates 14-35
Probing operation 14-36
Feature based machining 14-37
Rule–based operations for features 14-37
Feature recognition 14-38
Feature mapping 14-39
Machining Knowledge Editor 14-40
Standard machining knowledge content supplied with NX 14-41
Machining Feature Navigator filters 14-41
Collision Check and Incomplete status 14-42

Wire EDM 14-42
Tool Path Editor 14-43
ISV 14-44
General 14-44
CSE 14-45
Collision checking 14-46
Teamcenter Integration 14-46
Teamcenter Navigator with Manufacturing 14-46
Post Builder 14-47
UDE Editor 14-47
Virtual N/C Controller (VNC) 14-47
Localization 14-48
Other enhancements 14-49
Preview projects 14-49
What are preview projects? 14-49
IPW thickness containment curves 14-50
Divide by Holder 14-51

Automation 15-1

NX Open 15-1
Block Styler 15-1
Knowledge Fusion 15-1
KF TCE integration 15-1
KF versioning and deprecation 15-2
KF ICE best coding practices 15-2
Knowledge Fusion optimization enhancement 15-3
KF ICE Application packaging 15-3

NX to JT 16-1

New configuration option 16-1
Discontinued configuration options 16-1

Translators 17-1

3D PMI export to 2D Exchange 17-1
3D PMI export to DXF and DWG files 17-1

Enhancements in NX 5.0.x Maintenance Releases 18-1

Teamcenter Integration 18-1
Authorized data access 18-1
Advanced Search 18-6
Cloning multi-CAD items 18-8
Replacement component in an assembly retains data 18-9
Fully load data for current version of a part 18-9
Open Part File and Teamcenter Navigator columns are configurable 18-10

NX Essentials 18-11
Command Finder 18-11
Default names for new groups 18-12
Record and Play C# Journals 18-12
Select Scope in Sketcher 18-13
Print dialog box 18-13
Infer Edge Output 18-13
Design 18-14
Modeling 18-14
Assemblies 18-23
Documentation 18-25
PMI 18-25
Data Reuse 18-25
Teamcenter Classification objects in the Reuse Library 18-25
Machinery Library installation tool 18-26
NX Machinery Library 18-26
System Design 18-27
Die Engineering 18-27
Die Validation 18-29
Ship Design 18-31
Flexible Printed Circuit Design 18-39
PCBxchange 18-40
Digital Simulation 18-46
NX 5.0.1 Enhancements 18-46
NX 5.0.2 Enhancements 18-86
NX 5.0.3 Enhancements 18-99
Product Validation 18-105
Check-Mate 18-105
Manufacturing Design Tools 18-109
Weld Assistant 18-109
Mold and Die Tools 18-116
Manufacturing 18-118
Tool path editing — trimming 18-118
Tool Path Divide 18-121
Solid tool support in Teamcenter libraries 18-121
Generic Motion 18-122
Integrated Simulation & Verification (ISV) 18-124
Non Cutting Moves 18-125
Streamline scallop stepover 18-126
Automation 18-127
Knowledge Fusion 18-127
NXOpen API 18-129
Translators 18-129
NX to JT 18-129

Chapter

1 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 5, click on the entry in the Table of Contents titled **Enhancements in NX 5.0.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

2 Synchronous Modeling with synchronous technology

To meet the demands of design change, **Direct Modeling** is now **Synchronous Modeling** and is significantly enhanced with reliable and easy to use core technology and new comprehensive capabilities.

New to NX 6 is synchronous technology, an exciting innovation that enables freedom in design change not previously experienced. From finding and maintaining geometric conditions, to modification through dimensions, to modification by editing a section, to the distinct advantage of synchronous feature behavior independent of linear history, synchronous technology introduces a new way of modeling.

The essential goal of **Synchronous Modeling** remains that of presenting an approach for design change with emphasis on modifying the current state of a model without regard for how it was constructed, its origins, its associativity, or its feature history.

History–Free Mode

The new **History-Free Mode** creates features that do not accumulate a linear history (see History-Free Mode).

- New **Shell Body**, **Shell Face**, and **Change Shell Thickness** commands (see Adaptive Shell).
- New **Cross Section Edit** command (see Cross Section Edit).
- New **Merged Rib Faces** Selection Intent rule.

New face selection and interaction options

- **Face Finder** (see Move Face)
- **New Motion options** (see Geometric transform commands, Move Face, and Move Object)
- **Active Selection** (see Active Selection)
- **OrientXpress** (see OrientXpress)

- **Group Face** (see Group Face)
- New **Include Boundary Blends** face selection option.

Basic command set enhancements

- Enhanced Synchronous Modeling commands
- New Geometric transform commands
- New dimension commands: Linear Dimension, Angular Dimension, and Radial Dimension.

Reuse commands

- Reuse faces with new Reuse commands
- Enhanced Copying of faces)

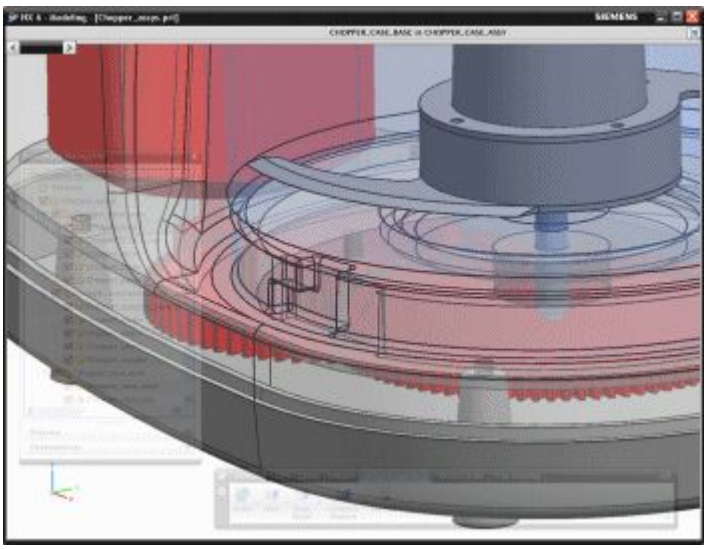
Core technology enhancements

- Support for topology change is greatly improved.
- Increased support for delete cases.
- Increased support for topology change in cases where blend faces overflow other blend faces.

Chapter

3 User efficiency

NX 6 contains a series of streamlined and configurable interface tools, and a new visual environment, crafted to maximize your workflow efficiency. These tools allow you to tailor NX settings to suit your design needs.



New visual environment

- **True Shading** for quick, realistic images of your model geometry.
- Full screen display with translucent dialog boxes (Translucent dialog boxes are available on Windows Vista only)
- De-emphasis setting options for work plane and work part

Tools that accelerate your workflow efficiency

- Selection MiniBar
- Customizable radial toolbars
- **Resource Bar** toolbar
- Toolbar Manager for full screen display
- List options displayed as short cut buttons in dialog boxes

Streamlined Interfaces

- Revised color palette with new **Color** dialog box
- Direct access to background color settings from **Preferences→Edit Background** dialog box
- Enhanced grid functionality with new **Grid and Work Plane** dialog box

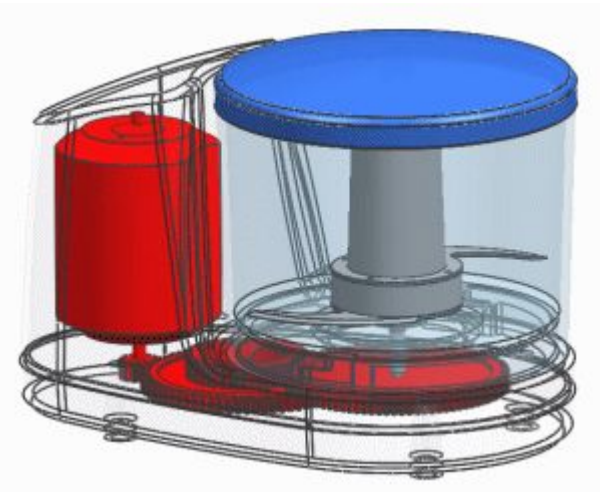
Chapter

4 Designing in an assembly context

Most products today are designed as assembles, not individual parts. Each part model must properly interface with every other part in the entire product assembly. NX 6 has several enhancements that work together to accelerate the workflows of designing part models in the context of a product assembly.

Assembly display

The work part in an NX 6 assembly is emphasized in a more natural way. The color of the other parts is de-emphasized, so that they fade into the background but are still recognizable.



The new Full Screen display allows you to use more of the screen area to work with complex assemblies. The **Assembly Navigator** and the Toolbar Manager can be semi-transparent so you can see the assembly behind them.

Modeling in an assembly context

NX 6 provides new methods of modeling within an assembly context to streamline the workflows of designing inter-related parts in a product assembly.

The traditional methods of sharing information from one part to another include:

- Interpart linked expressions

- **Copy** and **Paste**
- **WAVE Geometry Linker**

All of these methods are still available in NX 6 to reuse information between parts. These methods can still be used with or without creating permanent associative links.

New in NX 6, many common **Modeling** commands allow you to directly select geometry from other parts in the assembly. These enhancements are visible as two additions to the Selection bar.

Selection Scope

The new **Selection Scope** option on the Selection bar allows to you define if you want to select geometry only from the work part, components of the work part subassembly, or from the entire assembly. These selections are available within any specific command that includes this functionality.

Create Interpart Link

You can choose whether or not to create an interpart link when you select geometry from a part other than the work part. To create a permanent associative link, activate the **Create Interpart Link** button before selecting the geometry. If this button is inactive, this indicates that the creation of an associative link is not an option for the specific command or the particular selection option of that command. Do not activate this button if you want to select geometry from other parts in the assembly without creating a permanent link.

Sketching in an assembly context

The new Selection bar options described above are also available in some sketching commands.

For example, to create a sketch in an assembly context, use the **Selection Scope** option **Entire Assembly** to select a sketch face outside the work part. Use the **Create Interpart Link** button when selecting the sketch face to create an associative link.

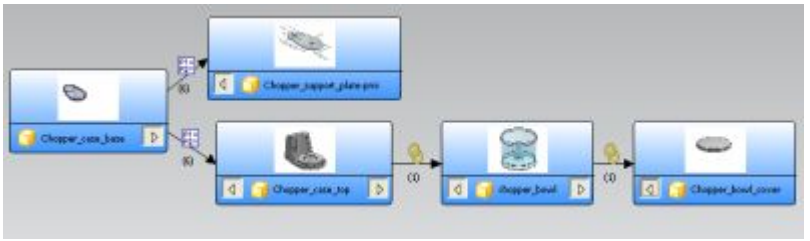
To constrain a sketch to curves in another part, project the curves into the sketch using these same options to create associatively linked curves. You can then create sketch constraints to these projected curves.

Facetted models

You can select geometry from other assembly components even if you display assembly components using lightweight facetted reference sets or JT files. When you select geometry to work in the assembly context, NX 6 automatically fully loads a component if it needs to.

Relations Browser

The associative links created in the work part are visible in the **Part Navigator**. To see the complete picture of relationships between parts in the assembly, use the new **Relations Browser**.



Workflow examples

These are some typical examples where you may use this new capability.

- Use a solid body in one part as an associative tool to unite, subtract, or intersect with a target body in the work part.

  In this workflow, use one of the Boolean commands with the **Selection Scope** option **Entire Assembly** and with the **Create Interpart Link** button activated.

- Use a surface in one part as an associative trimming tool with a target body in the work part.

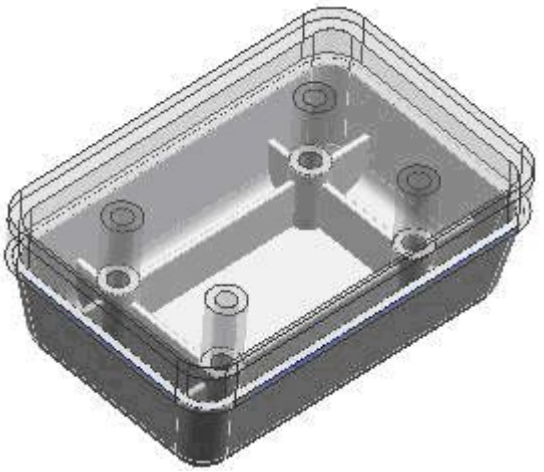
  In this workflow, use the **Trim Body** command and select the surface as the tool using the same Selection bar options.

- Create a mating interface or bolt pattern in the work part to match another part in the assembly.

  In this workflow, use the **Extrude** command and directly select the curves or edges to extrude. Set the **Selection Scope** option to **Entire Assembly** before selecting the section to extrude.

- Extrude a feature in the work part to touch another part in the assembly.

  In this workflow, select a face from the part as the extrude limit using the **Until Selected** option of the **Extrude** command.



- Use layout sketches for top-down design, such as to define mating interfaces and control volumes.

  In this workflow, extrude sections into the work part using the **Selection Scope** option **Entire Assembly**, with **Create Interpart Link** activated before selecting the sections to extrude. You can create the layout sketch in the parent assembly, or in a separate file in the assembly.

- Create sketched features in the work part to mate with features in another part.

  In this workflow, project curves into the sketch using the **Entire Assembly** and **Create Interpart Link** options to create associatively linked curves. Create constraints to these projected curves.

Chapter

5 Teamcenter Integration for NX

Non-geometric components in NX assemblies

What is it?

Non-geometric components (NGC) can now be displayed in **Assembly Navigator**. The assembly structure shown in the **Assembly Navigator** can now match what is shown in the drawing parts list and Teamcenter PSE.

An NGC does not need to have a dataset. If the item has a unit of measure defined, it is considered an NGC when the component is added to NX.

You can designate any component as non-geometric on the Assembly page of the **Component Properties** dialog box. Once you do this, the component is no longer displayed in the NX graphics window.

You can control whether you see NGCs in the **Assembly Navigator** by right-clicking and choosing the **Include Non-Geometric Components** filter.

The following also apply:

- You cannot make an NGC the displayed part or the work part.
- Many assembly commands are not valid for use with an NGC and are blocked.
- You cannot set an NGC as a reference-only component.
- When a parts list is created, the NGC items are also included in the parts list.

Why should I use it?

You can more accurately control the assembly structure and attribute values of an assembly in NX.

Where do I find it?

Application	Teamcenter Integration
Resource bar	**Assembly Navigator**

Parts list

What is it?

The parts list functionality is enhanced to provide more robust functionality and increased data integrity.

The underlying design of the parts list has been modified so that the parts list always reflects the changes made in Teamcenter PSE and the **Assembly Navigator**, but changes made in the parts list are no longer reflected in the assembly structure in the **Assembly Navigator** or Teamcenter PSE. The structure in the **Assembly Navigator** must be updated to get a corresponding change in the parts list.

Changes you make in the parts list are considered local changes in the Drafting application. When a cell for an item in the parts list is changed, such as quantity, the quantity is not changed in the **Assembly Navigator** or Teamcenter PSE.

Specific cells in the parts list can be locked so they cannot be modified and the parts list setup can also be locked so it cannot be modified.

Note

The **Callout** column in the parts list is a key field used by default for packing and corresponds to the sequence number in Teamcenter PSE. To ensure that multiple occurrences of the same component appear on the same line of the parts list, it is recommended that you change the Teamcenter preference **PS_new_seqno_mode** to **existing**. If you do not change this preference, turn off the **Callout** column as a key field. This will allow similar items to pack (by part number) in the parts list.

Why should I use it?

You gain full alignment between Teamcenter product sturucture and the drawing parts list, while maintaining the flexibility to edit parts list data directly when needed.

Where do I find it?

Application	Teamcenter Integration and Drawing
Resource bar	**Assembly Navigator**
Menu	**Insert→Parts List**

Additional columns in Assembly Navigator

What is it?

Additional columns are added to the **Assembly Navigator** to provide more information about the status, history, and state of an item. The following new columns are added:

TC Information This column indicates when a subassembly or component has been updated in Teamcenter by another user. When a new version or revision is available, a warning icon is displayed in the column for that part even if you already have the item opened in the session. If you have the latest version, the column is blank.

Tooltips are provided that let you know if a new version or revision exists, or if the BVR for an assembly has been modified. It also states the user that updated the item. For example:

> <username> has created a newer revision of this part after you opened it.

If a node in the **Assembly Navigator** is unloaded, the column shows out of date for a newer revision. It does not show that for a newer version since only the latest version would be loaded.

You can update the status for a displayed assembly with the **Refresh Teamcenter Information** command.

Checked Out By This column shows who has the component checked out. It shows the user of the session if that is who has checked out the component. If the component is checked in, the column is blank.

Projects This column lists the project to which an item is assigned. If it is assigned to more than one project, all the projects are listed.

Status This column shows the current status assigned to the Teamcenter dataset. If more than one status is assigned, they are all listed. If it has no status, the column is blank.

Load State This column provides the load status for the item: unloaded, partially loaded, or fully loaded

Modifiable This column uses the information contained in other columns to determine whether you have write access to the part. A checkmark icon, none icon, or question mark icon indicates whether you are able to modify the part. If the part is not modifiable, a tool tip explaining the reason is provided. The following columns are used as input:

- Read-only
- Modifiable
- Checked Out By
- Status
- TC Information

Quantity This column provides the quantity of the item within the assembly.

Callout This column shows the Teamcenter sequence number of the component.

Where do I find it?

Application	Teamcenter Integration
Resource bar	**Assembly Navigator**

Refresh Teamcenter Information

What is it?

You can check the status of each component in the assembly and update several columns with the **Refresh Teamcenter Information** command in the **Assembly Navigator**. Right-click a column to access the command and update the following columns:

- TC Information
- Checked Out By
- Status
- Modifiable
- Read-only
- Projects

- Precise Structure

  Note

  The **Update Precise Structure** command is replaced with the **Refresh Teamcenter Information** command.

If you have modified a part that has also been modified by another user, you are prompted with an informational message.

Where do I find it?

Application	Teamcenter Integration
Shortcut menu	**Assembly Navigator**® right-click on a column

Reset columns

What is it?

The configuration of columns in **Teamcenter Navigator** and **Assembly Navigator** can be modified, as required. You can now reset the columns to revert to the default state with the **Reset to Default** button on the configuration dialog box.

The default is defined by the preferences that control configuration of columns, such as the Teamcenter preference **TC_NX_TCNavigator_Column_Config**. If no column preferences are defined, then the default is the configuration that is delivered with NX.

Where do I find it?

Application	Teamcenter Integration
Resource bar	**Teamcenter Navigator** and **Assembly Navigator**

Quantity in an assembly

What is it?

You can specify the quantity of a component in an assembly by entering the number in the **Quantity** box on the Assembly page.

A **Quantity Type** list is also provided on the Assembly page that lets you select **As Required (A/R)** or **Number** to specify the quantity in the **Quantity** box.

Note

If a unit of measure (UOM) is defined for the item, the quantity is a real number. If no UOM is defined, the quantity is an integer.

Why should I use it?

You can enter the quantity of an item for an assembly.

Where do I find it?

Application	Teamcenter Integration
Shortcut menu	**Assembly Navigator**® right-click a component® **Properties**
Location in dialog box	**Component Properties** dialog box® Assembly page® **Quantity** box

Reopen parts modified by other users

What is it?

From the **Reopen Part** dialog box, you can now reopen parts you already have open that were modified by other users.

When you select the **List Parts Modified by Other Users** checkbox, the following is applicable:

- Parts that were modified by other users since you opened them in the current session are listed.

  Note

  Parts that are modified but have not been loaded are not listed.

- Parts that are partially loaded are listed.
- Users that have modified the part are listed in the **Modified By** column.
- The **Reopen All Parts** button closes and reopens all parts that have been modified.

Where do I find it?

Application	Modeling
Menu	**File→Close→Reopen Selected Parts**

Assign items to projects

What is it?

You can assign Items, Item Revisions, and datasets to a project and also remove those from a project at the following locations within NX:

Note

This is available with Teamcenter 2007.

- In **Teamcenter Navigator**, when you right-click an item and choose **Projects**.
- In **Assembly Navigator**, when you right-click a component and choose **Projects**.
- In the **File New** dialog box, when you click the **Projects** button.
- In the graphics window, when you right-click the displayed item and choose **Projects**.
- In the **File Save As** dialog box, when you click the **Projects** button.
- In the **Name Parts** dialog box, when you click the **Projects** button.
- In **File→Utilities→Projects**, when the work part is the active part.

Note

You can assign an item to multiple projects. You can also select multiple items to assign to a project.

A **Projects** column is added to **Teamcenter Navigator** and **Assembly Navigator**. A **Projects** top level node is added to **Teamcenter Navigator**. The **Projects** node shows nodes for the project hierarchy with project folders in each project node. A project folder shows the contents of the project, such as Items, Item Revisions, and datasets.

Why should I use it?

You can assign items to a project to help in organizing and maintaining data for the project.

Where do I find it?

Projects

Application	Teamcenter Integration
Resource bar	**Teamcenter Navigator** and **Assembly Navigator**

Menu **File→New/Save As**

Teamcenter columns in NX

What is it?

You can display Teamcenter columns in **Teamcenter Navigator**, **Assembly Navigator** and the **Open Part File** dialog box. The Teamcenter columns are available with the other NX columns but are not displayed unless specifically selected, or defined in the column configuration preferences.

The following Teamcenter preferences allow the additional columns to be available in NX:

Note

These are site level preferences.

- **TC_NX_additional_columns**

  This preference makes available columns for workspace objects.

- **TC_NX_additional_item_columns**

  This preference makes available columns for items.

- **TC_NX_additional_dataset_columns**

  This preference makes available columns for datasets.

The **Teamcenter Navigator** and **Open Part File** dialog box columns can be configured at the user, group, or role level with the following column configuration preferences. The preferences enable display of the specified columns automatically. The columns are specified by listing the column name followed by a comma and must be made available by activation of the preferences above. They are displayed after the default columns delivered with NX.

- **TC_NX_TCNavigator_Column_Config**

  This preference lets you choose the columns that are displayed for **Teamcenter Navigator**.

- **TC_NX_File_Select_Column_Config**

  This preference lets you choose the columns that are displayed for file selection dialog boxes, such as **Open Part File**.

The **Assembly Navigator** columns can be configured at the user, group, or role level with the following preference:

- **TC_NX_ANT_Column_Config**

  This preference lets you choose the additional Teamcenter columns that are displayed after the **Assembly Navigator** columns.

  Note

  In NX, you can change the columns by right-clicking on a column and selecting **Columns→Configure**.

Why should I use it?

You can include Teamcenter information by adding Teamcenter columns and manipulate the display of columns as applicable at your site.

Where do I find it?

Application	Teamcenter Integration
Resource bar Menu	**Teamcenter Navigator** and **Assembly Navigator** **File→Open**

Freeze Column

What is it?

You can use **Freeze Column** to prevent columns from scrolling horizontally in **Teamcenter Navigator**. This allows you to use certain attributes to keep track of items, such as name or revision, while still scrolling to see other attributes of the items.

Note

This also applies to **Assembly Navigator**.

When you freeze columns, a line is drawn to the right of the column where you selected the freeze. Columns to the left of the line are frozen; columns to the right of the line are scrollable.

Why should I use it?

You can keep specific identifying columns fixed while checking the other columns.

Where do I find it?

Application	Teamcenter Integration
Shortcut menu	**Teamcenter Navigator** and **Assembly Navigator**® right-click on the column heading you want to use as the dividing line® **Freeze Column**

Projects in Teamcenter Navigator

What is it?

You can see the contents of projects in **Teamcenter Navigator**. The **Projects** top level node shows nodes for the Teamcenter projects. Each project shows the contents of the project, such as Items, Item Revisions, and datasets.

Why should I use it?

You can display the contents of projects.

Where do I find it?

Application	Teamcenter Integration
Resource bar	**Teamcenter Navigator**

Item name

What is it?

NX now shows the **Item Name** in addition to the **Item Number** by default in the **File New**, **File Open**, **Save As**, and **Name Parts** dialog boxes. Also, both the **Item Name** and **Item Number** are shown in the window title bar.

You can also set the customer default **Use Item Name Instead of Item Number** to display **Item Name** instead of **Item Number** in the rest of the NX interface not covered by the dialog boxes listed above. This setting does not affect the dialog boxes listed above. For example, **Item Name** is used instead of **Item Number** in the object column of **Teamcenter Navigator**.

Why should I use it?

You can use the item name in addition to the item number to further identify an item. You can also set the customer default if you prefer to see the item name instead of the item number in the rest of the NX interface.

Where do I find it?

File New, **File Open**, **Save As**, **Name Parts** dialog boxes.

Use Item Name Instead of Item Number customer default:

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Teamcenter Integration for NX→General→**General page

Import assembly

What is it?

When you import an assembly into Teamcenter and click **Add Assembly** or **Add Part** on the **Import Assembly** dialog box, you can now select multiple files to import from the file selection dialog box.

An **Add From Folder** button is added to the **Import Assembly** dialog box that launches a folder selection dialog box. From there you can select the folder that contains the items to add to the import list. All the .prt files contained in the folder are added to the list. Assemblies that cannot be loaded with the current load options settings are listed after all the items in the folder have been processed. The contents of sub-folders are not added to the import list.

Why should I use it?

You can select multiple files to import at once and easily populate the list of importable items.

Where do I find it?

Application	Teamcenter Integration
Menu	**File→Import Assembly into Teamcenter**

Teamcenter Integration commands moved

What is it?

Teamcenter Integration and its submenus are removed from the **Tools** menu. The commands are relocated as follows:

Command in Tools® Teamcenter Integration	New Location
Import Assembly	**File→Import Assembly into Teamcenter**
Export Assembly	**File→Export Assembly outside Teamcenter**
Save Precise Assembly	**File→Save Precise Assembly**
Save Outside Teamcenter	**File→Utilities→Save Outside Teamcenter** (must be turned on)
Refile Pattern Parts	**File→Utilities→Refile Pattern Parts** (must be turned on)
Save All Parts	**File→Force Save All**
Add Alternate ID's	**File→Add Alternate ID's**

Command in Tools® Teamcenter Integration	New Location
Revision Compare	**File→Utilities→Prepare for Compare** (must be turned on)
Manage Pending Components	**Assemblies→Components→Manage Pending Components**
List Locked Parts	**File→Utilities→List Checked-Out Parts** (must be turned on)
Lock Modified Parts	**File→Utilities→Check-Out Modified Parts**
Lock/Unlock Parts	Moved to **Teamcenter Navigator**. Right-click component® **Check In** and **Check Out**
Lock/Unlock Remote Parts	Moved to **Teamcenter Navigator**. Right-click component® **Remote Check In** and **Remote Check Out**

Note

To turn on a menu selection, choose **Tools→Customize** and then click the **Commands** tab. Right-click **File** in the **Categories** list, and then right-click **Utilities** in the **Commands** list and choose **Add or Remove Buttons**. You can then select the menu items you want.

Why should I use it?

The commands are easier to find in the Teamcenter Integration application.

Where do I find it?

Application Teamcenter Integration

Import part with validation options

What is it?

While importing parts to Teamcenter, you can validate part files using validation rules. The new **Validation** tab in the **Import Assembly** dialog box provides options for setting up and running a validation.

Note

- A file must already exist and contain rule definitions. It can be retrieved either from Teamcenter or from native NX (file system).
- NX CheckMate kit must be installed.
- The options on the Validation page correspond to the options available for the command line tool **ug_clone** (introduced in NX 5).

Validation Options

The following options are available on the Validation page:

Validation Mode — Specifies how to import validation results during import of NX parts.

The available options are: **Off, Import from Part, Run Validation, Run Validation (Hybrid)**.

The **Run Validation (Hybrid)** option allows you to optimize the validation performance by avoiding re-run validation against the part files which have previously stored results.

Validation Rule Set File — Specifies the file containing the validation rules and whether it is in Teamcenter or native NX (on the file system).

Treat Warning as Pass — Indicates whether the **Passed with Warning** status in CheckMate should be accepted.

Treat Outdated as Pass — Indicates whether the **Passed but Outdated** status in CheckMate should be accepted.

Exceptions — Displays the **Validation Exceptions** dialog box where you can select options as exceptions to certain part files. The options **Validation Mode, Validation Rule Set File, Treat Warning as Pass**, and **Treat Outdated as Pass** are the same options available in the **Validation** tab.

Abort Import on Fail — Stops the import of parts if any of the validation rules are not satisfied.

Why should I use it?

You can check the data quality of parts from a supplier against your company standard before importing them into Teamcenter. In addition, Check-Mate results from parts validated in native NX can be automatically mapped to validation objects when the parts and assemblies are imported into the Teamcenter database.

Where do I find it?

Application	Teamcenter Integration
Menu	**File→Import→Assembly into Teamcenter**
Location in dialog box	**Validation** tab

Template setup

What is it?

Changes are made to the **File New** dialog box template setup script to provide more consistency during subsequent releases.

- The name of the template setup script is changed from **nx5_template_setup.bat** to **tcin_template_setup.bat**.
- The template parts are now located in the **NX Templates** folder instead of the **NX5 Templates** folder in My Teamcenter.
- The action for the **–default_a** option of the **ug_clone** script when running the setup script is now **new_revision** instead of **overwrite**. This is also displayed in the command window when the script is running.

Why should I use it?

The script is now generic to all NX versions.

Where do I find it?

tcin_template_setup.bat.

Path	**%UGII_ROOT_DIR%\templates\sample**

Chapter

6 NX Essentials

Full screen display

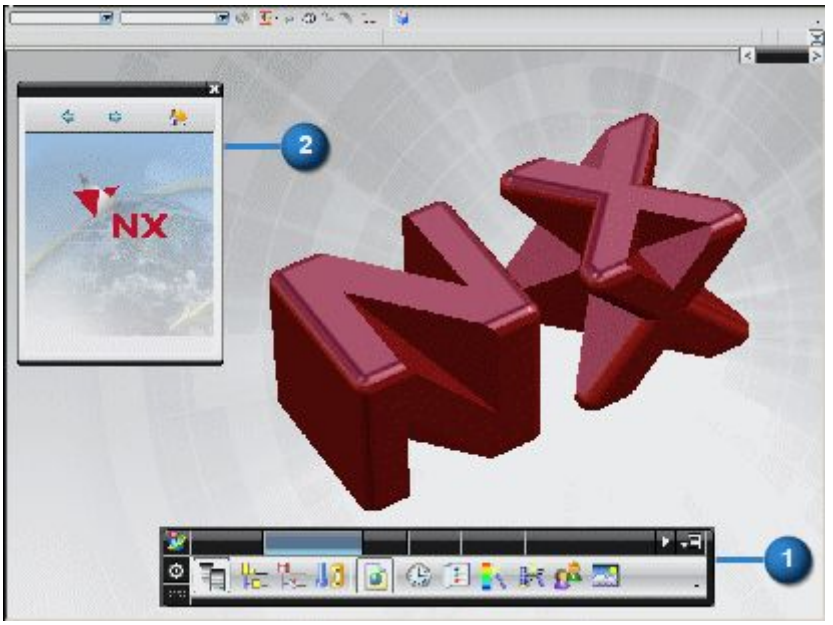
What is it?

You can now maximize the NX main window using the **Full Screen** option. With this option you have more screen space in the graphics window, as the menu bar, toolbars, Resource bar and Selection bar are all removed and made available through alternate means.

With a full screen display, you can access:

- The menu bar and all NX commands from a new Toolbar Manager.
- The Resource bar functionality from the new **Resource Bar** toolbar.
- The Selection bar options from the new Selection MiniBar.
- Toolbar commands from customized radial toolbars.

Full screen display remains in effect, even when no part is displayed, until you turn it off and return to the standard display.



Full screen display

1 — Toolbar Manager with **Resource Bar** toolbar

2 — Resource bar window

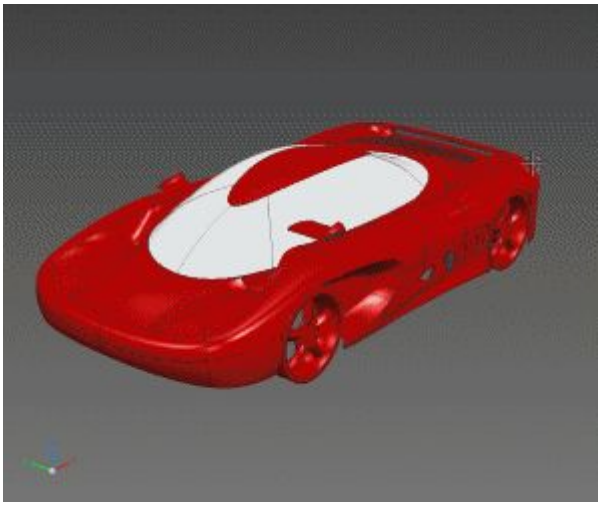
Why should I use it?

You have a larger graphics window for ease of modeling, and you can design more efficiently as the Toolbar Manager, Selection MiniBar, and customizable radial toolbars all help to accelerate your workflow.

Where do I find it?

Menu	To turn full screen display on: **View→Full Screen** To turn full screen display off: Toolbar Manager® menu bar button ® **View→Full Screen**
Keyboard accelerator	Alt+Enter
Location in graphics window	Full screen button to the right of the Cue/Status line

True Shading



What is it?

True Shading provides you with the ability to create realistic visual effects in your NX models. The effects include shadows, reflections, lighting, and materials, and are intended to be used as visual aids as you model a part or configure an assembly. Previously, you could do this only if you had a **Studio Visualize** license.

True Shading includes many standard capabilities provided in both Shaded and Basic/Advanced Studio rendering styles. You can use **True Shading** to:

- Apply a material finish to all objects.
- Apply a material finish to selected bodies and faces.
- Display default, inherited, and custom shaded background colors and images.
- Show floor reflection, floor shadow, and floor grid.
- Apply any of several light collections, or customize your active light collection.

Why should I use it?

Use True Shading functionality to enhance the display and provide a level of realism to your NX model.

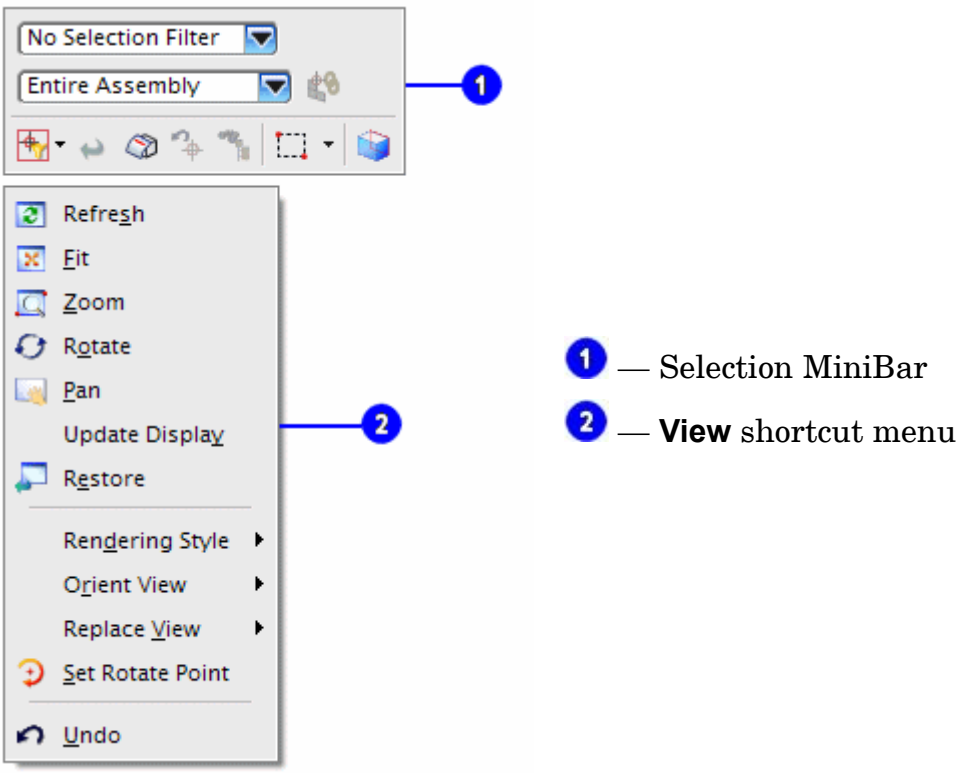
Where do I find it?

Application Toolbar	Gateway or Modeling **True Shading® True Shading**
Menu	**View® Visualization® True Shading Editor**

Selection MiniBar

What is it?

You can display a compact version of the Selection bar in the graphics window whenever the **View** shortcut menu is in use. The Selection MiniBar is available for both full screen display and standard display.



Why should I use it?

The Selection MiniBar increases your workflow efficiency by bringing the selection functionality directly to your cursor location thus reducing required mouse movement.

Where do I find it?

Graphics window	Right-click, or hold Ctrl and right-click when the cursor is over geometry.

Selection MiniBar On/Off option

Menu	**Tools→Customize**
Shortcut menu	Right-click in the toolbar area →**Customize**
Location in dialog box	Layout page ® **Show Selection MiniBar**

Customizable radial toolbars

What is it?

Customizable radial toolbars are application-specific radial toolbars that can be customized for each NX application. You can configure one or more of your

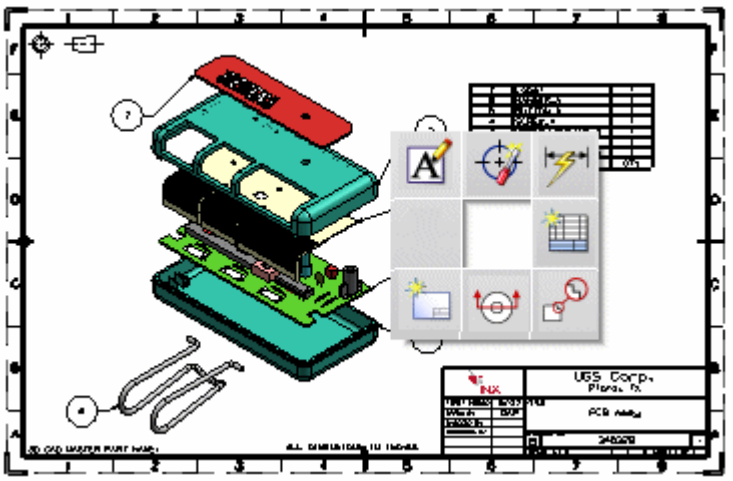
radial toolbars from the **Customize** dialog box by dragging the commands you want to display into the radial toolbar placeholder.

You can:

- Include up to eight commands in each radial toolbar.
- Access each radial toolbar from one of the three mouse buttons.

When you hold Ctrl+Shift and click a mouse button, the radial toolbar associated to that button appears.

The following figure shows an example of a customized radial toolbar in the Drafting application.



You can use customizable radial toolbars in either a full screen display or a standard display.

Why should I use it?

Being able to display your most used commands in radial toolbars increases your workflow efficiency by bringing the commands directly to your cursor location.

Where do I find it?

Graphics window	Hold Ctrl+Shift and click the mouse button to which your toolbar is mapped.

Customize options:

Menu	**Tools→Customize**
Shortcut menu	Right-click in the toolbar area→**Customize**
Location in dialog box	Toolbars page® **Radial 1**, **Radial 2**, or **Radial 3**

Resource Bar toolbar

What is it?

The Resource bar is now available in a toolbar format, with buttons for accessing the navigators, browsers and palettes.

In standard display, you can use either the **Resource Bar** toolbar or the Resource bar, but not both.

In full-screen display, the Resource bar is hidden to increase available screen space in the graphics window. Resource bar functionality is available in the **Resource Bar** toolbar in the Toolbar Manager.

From the **Resource Bar** toolbar, you can:

- Display a single docked or undocked window for the Resource bar navigators, browsers or palettes.
- Display multiple undocked windows for navigators, browsers or palettes.
- Customize the buttons displayed on the **Resource Bar** toolbar.

Why should I use it?

You can customize the **Resource Bar** toolbar and display it in either full screen mode or standard mode. You can also place the **Resource Bar** toolbar buttons on a customizable radial toolbar, providing access to Resource bar functionality directly at your cursor location.

Where do I find it?

Display **Resource Bar** toolbar in standard display

Menu	**Preferences→User Interface**
Location in dialog box	Layout page® **Resource Bar→Display Resource Bar**

Display **Resource Bar** toolbar in full screen display

Menu	**Tools→Customize→Resource Bar**
Shortcut menu	Right-click in the toolbar area ® **Resource Bar**

Display **Resource Bar** customer defaults

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Gateway→User Interface**→Layout page® **Display Resource Bar**

Toolbar Manager

What is it?

The Toolbar Manager is a compact toolbar that provides access to menus, all the currently active toolbars, and the Resource bar. All typical NX workflows are supported from the Toolbar Manager.

> **Note**
>
> The Toolbar Manager is only available when you run NX with a full screen display.

From the Toolbar Manager, you can:

- Display toolbars available in the active Role from individual tabs.
- Change the displayed toolbar by cycling through the tabs.
- Add or remove toolbars.
- Press and release Alt to quickly display the active toolbar as a floating palette of commands at your cursor location.
- Right-click on an inactive toolbar command to display a floating palette of commands for that toolbar.
- Access the menu bar from a single button.

You can also:

- Re-order the Toolbar Manager tabs.
- Customize the visible toolbars independent from what is visible in the standard display, and save your choices in an NX Role.
- Minimize the Toolbar Manager to a single button to further maximize the graphics window.
- Snap the Toolbar Manager to the top or bottom of the graphics window.

Why should I use it?

The Toolbar Manager provides rapid access to the menu bar, toolbars and commands while in a full-screen display. It also provides enhanced modeling efficiency by increasing the available space in the graphics window and reducing mouse movement needed to access different NX functions.

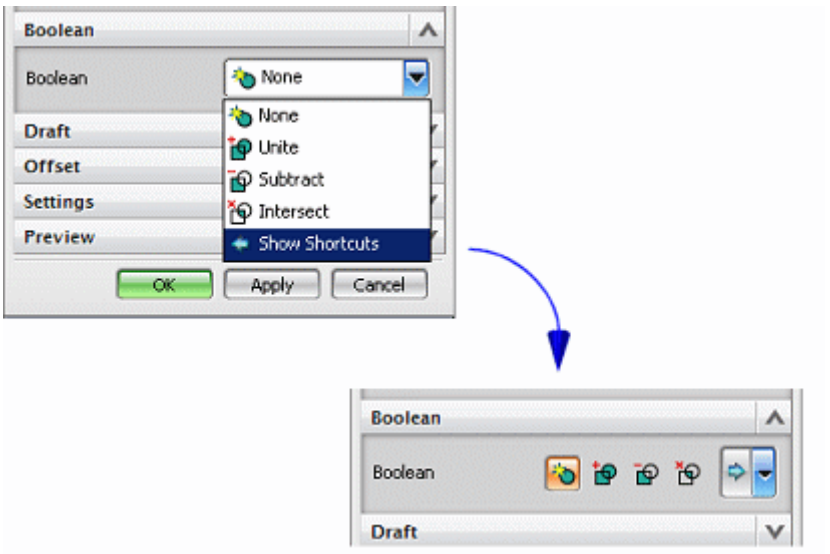
Where do I find it?

Prerequisite	You must be in a full-screen display to use the Toolbar Manager.
Location in the graphics window	The Toolbar Manager is immediately presented when you enter the full-screen display.
Keyboard accelerator	Press Alt to bring a floating palette of commands to your cursor location.

Shortcuts

What is it?

In many dialog boxes, you can now display listed options as buttons using the new **Show Shortcuts** option. The button you click last is, by default, the active button the next time you return to the same dialog box.



Boolean list as shortcut buttons

Why should I use it?

Having options available as buttons instead of listed items reduces the mouse movement needed to select them.

Where do I find it?

Location in dialog box	**Show Shortcuts** is available for option lists

De-emphasis settings

What is it?

New **De-emphasis Settings** options in the **Visualization Preferences** dialog box give you greater control over the appearance of objects you want to de-emphasize, like non-work parts in an assembly or objects not on the work plane.

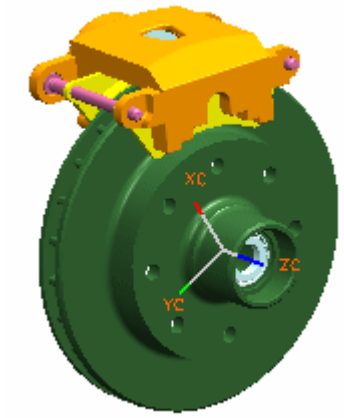
With these de-emphasis options you can:

- Set the blending color to any color in the part color palette.
- Set the blend percentage value.

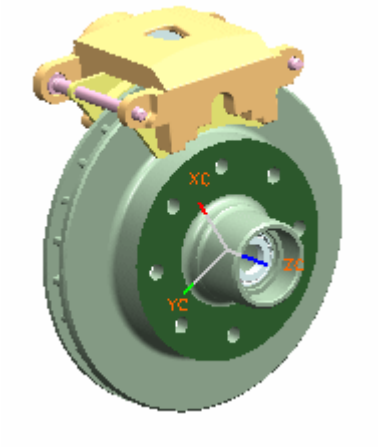
Note

Color blending is applied to geometric objects only.

The example below shows how the **De-emphasis Settings** affect the display of objects not on the work plane.



No objects de-emphasized



Objects not on workplane de-emphasized

The **De-emphasis Settings** options replace the **Work Plane Emphasis** and **Assemblies Work Part Emphasis** options that were previously available in the **Session Settings** group of the **Visualization Preferences** dialog box.

Where do I find it?

Application	Gateway
Menu	**Preferences→Visualization**
Location in dialog box	Color Settings page® **Part Settings** group →**De-emphasis Settings** subgroup ® **Blend Color** and **Blend Percentage**

Blend settings customer defaults

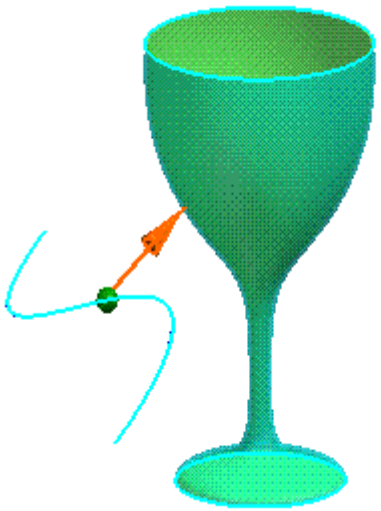
Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Gateway→Visualization→** Color Settings page

Vector Constructor

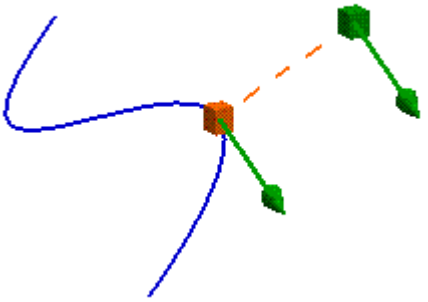
What is it?

The **Vector Constructor** tool has several enhancements. You can now:

- Provide a through point when creating a vector normal to a non-planar face using the **Face/Plane Normal** vector type. If the point is not on the face it will be projected to the face using the shortest distance.
- Provide a point when creating a vector tangent to a curve using the **On Curve** vector type. If the point is not on the curve it will be projected to the curve using the shortest possible distance.
- Create a vector from an expression using the new **By Expression** type.
- Create a vector derived from the view plane using the new **View Direction** type.
- Select a B-Surface when using the **Face/Plane Normal** type.



Vector normal to a non-planar face through a point



Vector tangent to a curve at a point

The following types are changed:

- The **Face Normal** and **Plane Normal** types are combined into one **Face/Plane Normal** type .
- The **Edge/Curve** type and **Datum Axis** type are combined into one **Curve/Axis Vector** type .

Why should I use it?

You can now quickly create vectors at specified points, and by view direction or expressions.

Where do I find it?

These options are available wherever the **Vector Constructor** tool is used.

Example

Toolbar	**Feature→Extrude**
Menu	**Insert→Design Feature→Extrude**
Location in dialog box	**Specify Vector** → click **Vector Constructor** for the **Vector** dialog box, or select an option from the Vector list

CSYS Constructor

What is it?

The CSYS Constructor tool has several enhancements. You can now:

- Create an associative, dynamic offset CSYS relative to a selected CSYS using the **Dynamic** creation method.
- Create a CSYS by specifying an origin, a Z-axis and an X-axis.
- Create a CSYS by specifying an origin, a Z-axis and a Y-axis.
- Specify a CSYS axis that is:
  - Derived from a surface vector normal at a point projected from the CSYS origin onto the surface.
  - Derived from a vector tangent to a curve at a point projected from the CSYS origin onto the curve.
- See a preview of an offset CSYS while it is being created.

Where do I find it?

These options are available wherever the CSYS construction tool is used.

Example

Application	Modeling
Toolbar	**Feature→Datum CSYS**
Menu	**Insert→Datum/Point→Datum CSYS**

OrientXpress tool

What is it?

A new tool is now available to quickly identify a principle axis, a principle plane, or both. The OrientXpress tool is displayed whenever you must specify a direction and/or orientation in order to complete a specific command.

OrientXpress tool

Comprised of a set of orthogonal axes and planes, OrientXpress is paired with other NX commands that require an orientation input, such as Synchronous Modeling's Linear Dimensions. When these commands need an axis or plane specified, you can use traditional tools like the **Vector** constructor, or you can specify the orientation input by selecting an axis or plane directly from the OrientXpress tool.

Why should I use it?

The OrientXpress tool increases your workflow efficiency by placing orientation controls directly in the graphics window. You no longer need to access separate controls or dialogs to specify principle planes or directional vectors.

Where do I find it?

This tool is presented whenever a command or another construction tool, such as the **Vector** constructor, requires an orientation input.

Example

Application	Modeling
Toolbar	**Synchronous Modeling® Linear Dimension**
Menu	**Insert® Synchronous Modeling® Dimension® Linear Dimension**

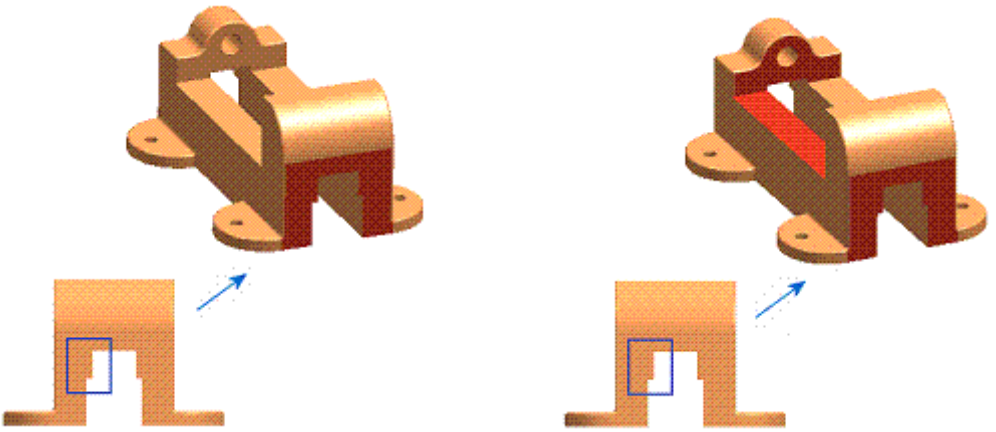
Allow multi-select of hidden faces

What is it?

You can now add or exclude hidden faces, bodies, and components from your lasso and rectangle multi-selections using the new **Allow Multi-Select of Hidden Faces** option.

You can select hidden faces that are within the selection boundary, as long as they are at least partially visible in the current orientation of the view.

Also, the **Mouse Gesture** and **Selection Rule** options from the **Selection Preferences** dialog box are now available on the Selection bar as **Multi-Select Gesture** and **Multi-Select Rule** list menus.



Selection with Multi-select turned off **Selection with Multi-select turned on**

Where do I find it?

Toolbar **Selection→Allow Multi-Select of Hidden Faces**

Preferences

Color dialog box

What is it?

The **Color** dialog box provides a revised color palette, including a range of contemporary colors in a new presentation style. In the **Color** dialog box, you now have:

- An increased selection of favorite colors.
- **Gray** and **Principle** color scales for ease of selection.
- **Vibrant**, **Neutral**, **Dark**, and **Light** color groups to assist in color identification.
- An increased range of diverse colors including contemporary color tones.

You can:

- Sort colors by Index Number order (that is, 1 through 216).
- Edit colors by right-clicking a color swatch.

Preferences for colors are now located at **Preferences® Color Palette**.

Why should I use it?

You can now customize object colors from a wide selection of modern and contemporary colors. This provides a more efficient, pleasing, and visually rewarding CAD environment in which objects are easily distinguishable from your NX background, resulting in improved ease of use.

Where do I find it?

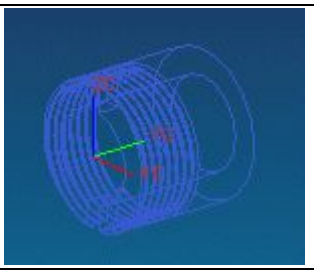
Menu	**Preferences**® **Color Palette** **Edit**® **Object Display**
Location in dialog box	**Edit Object Display** dialog box® Basic page® **Color**

Wireframe Contrast

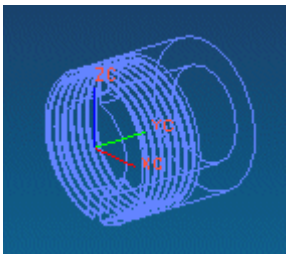
What is it?

You can use the **Wireframe Contrast** function to let NX automatically adjust the colors in your wireframe model for maximum contrast with your background color.

In the figure on the left, the blue model is difficult to see against the blue background. In the figure on the right, **Wireframe Contrast** applies a much brighter blue to the model so it is more visible.



Model without Wireframe Contrast



Model with Wireframe Contrast

Note

If you want the color of your model to be the same in both wireframe and shaded rendering styles, you should not use wireframe contrast.

Why should I use it?

Wireframe Contrast makes your wireframe model easier to see.

Where do I find it?

Prerequisite	Wireframe contrast can only be applied to wireframe models. It has no effect on shaded models.

Toolbar	**View** toolbar® **Wireframe Contrast**
Menu	**Preferences**® **Visualization**
Location in dialog box	**Line** tab® **Wireframe Contrast**

Grid and Work Plane

What is it?

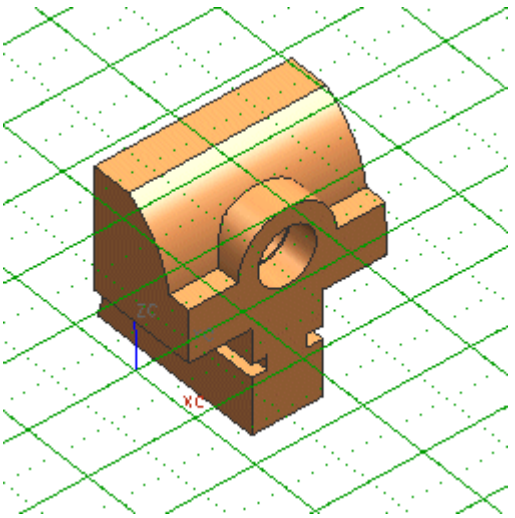
The **Work Plane Preferences** dialog box is now the **Grid and Work Plane** dialog box.

When you open the dialog box, the **Show** option is on by default, so the grid appears when you click **OK** or **Apply**.

You can now:

- Define grid spacing by specifying one number.
- Provide more divisions in your grid by specifying **Minor Lines Per Major** and **Snap Points Per Minor**.
- Define whether grid lines should be visible through objects.

Your grid spacing is now honored during operations such as zoom. Grids no longer resize during such operations.



Rectangular Non-Uniform grid with major and minor lines, on top of model

Where do I find it?

Application	Gateway and Modeling, primarily

Menu	**Preferences® Grid and Work Plane**
Location in dialog box	Grid spacing **Grid Size® Major Grid Spacing** The name of this option varies slightly, based on your grid **Type**.
	Optional divisions in the grid **Grid Size® Minor Lines Per Major** and **Snap Points Per Minor** The name of these options vary slightly, based on your grid **Type**.
	Display the grid **Grid Settings® Show**
	Show the grid through objects **Grid Settings® On Top**

Automatic model size preference for facets

What is it?

There is a new **Automatic** setting for your profile **Model Size**. When you use this setting, NX adjusts the profile by automatically switching between the **Small**, **Medium**, and **Large** settings to give you optional viewing characteristics for your model.

Why should I use it?

This option is useful when you work with faceted representations in a large model.

Where do I find it?

Menu	**Preferences® Visualization Performance**
Location in dialog box	**Large Model® Profiles® Model Size® Automatic**

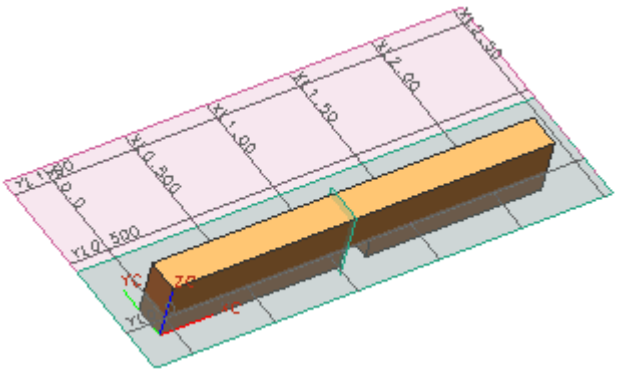
Datum plane grid

What is it?

You can place a grid directly on a datum plane or a plane of a datum CSYS with the new **Datum Plane Grid** command.

You can create or edit a datum plane grid in any application where you can create a datum plane, such as Modeling.

The following figure shows a datum plane grid. The grid's boundaries are expanded beyond the datum plane to make the grid labels more readable.



Datum plane grid

You can:

- Create multiple datum plane grids in your model.
- Rotate a datum plane grid within the plane it is located on.
- Create section curves where the grid intersects the model after you rotate the grid to the orientation you want.
- Use the model's work plane grid even when datum plane grids are visible.

Why should I use it?

A datum plane grid is a localized grid that helps you work on an object in your model. Grids typically provide the context of location and size to objects.

The ability to create multiple datum plane grids with different locations, orientations, and parameters lets you have grids that are optimized for individual objects in your model.

Where do I find it?

Application	Modeling and other applications where you can create a datum plane
Menu	**Insert® Datum/Point® Datum Plane Grid**
Shortcut menu	Right-click a datum plane® **Datum Plane Grid**

Exit and Close

What is it?

The **Exit** and **Close** commands are enhanced to enable you to immediately save modified files. The following table lists the buttons now available in the **Exit** dialog box, and in the **Close** dialog box when you do not select a specific save operation.

Exit dialog box		**Close** dialog box	
Button	Action	Button	Action
Yes — Save and Exit	Saves modified files and exits NX.	**Yes — Save and Close**	Saves modified files and closes them.
No — Exit	Exits NX without saving any changes to the files.	**No — Close**	Closes the files without saving any changes.
Cancel	Revokes the **Exit** command.	**Cancel**	Revokes the **Close** command.

Warning

In previous NX versions if you selected **Yes** while trying to close or exit the NX session after you made changes to a file, a warning would appear. Now, the default **Yes — Save and Exit** or **Yes — Save and Close** button automatically saves all modified files. Be careful to not inadvertently overwrite your files with any unwanted edits when you close files or exit NX.

Why should I use it?

You no longer need to cancel the **Exit** or **Close** command first in order to save your file.

Where do I find it?

Menu	**File→Close→Selected Parts** or **All Parts** **File→Exit**
NX Window	The close button

Hide objects immediately

What is it?

The new **Immediate Hide** command hides objects as soon as you select them in the graphics window.

Why should I use it?

Use this option when you want to hide objects with as few clicks as possible.

Where do I find it?

Toolbar	**Utility® Immediate Hide**
Menu	**Edit® Show and Hide® Immediate Hide**

View Section

What is it?

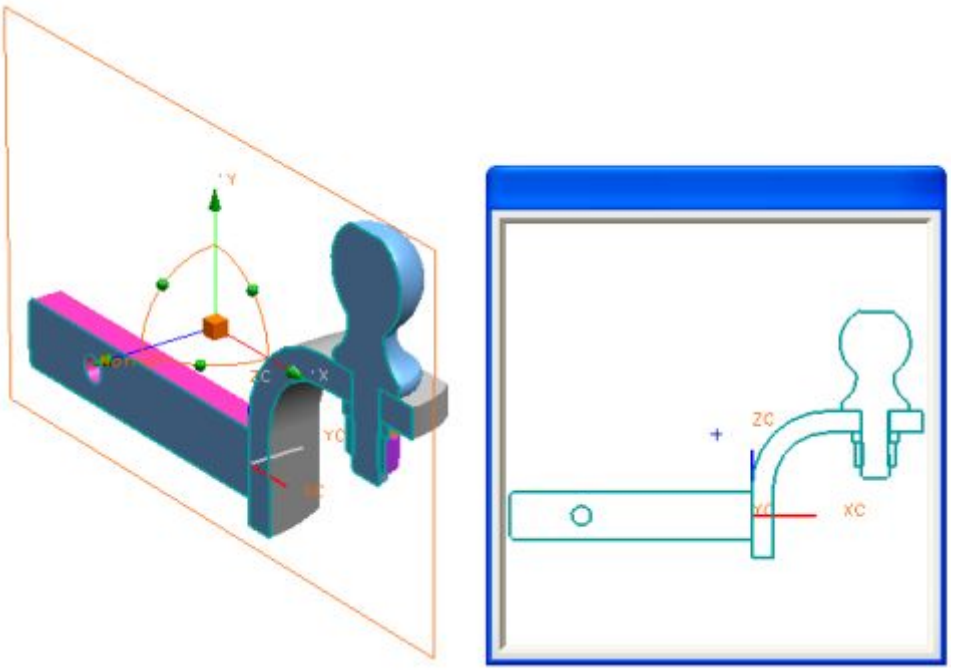
Use **New Section** to create cross section curves at various locations through all displayed parts.

You can:

- Have a series of sections, with optional specified spacing between each section.
- Display a cross section in a 2D view.
- Turn the cross section on or off in the graphics window using **Clip Work Section**.
- Use **Edit Work Section** to easily edit an existing cross section.

  Note

  Sections are listed in the **Assembly Navigator** under the **Sections** folder.



Dynamic Sectioning in 3D and 2D Viewer

Why should I use it?

Use the cross sections for visual inspection of parts; for example, to examine how components in an assembly interact with each other.

Where do I find it?

Toolbar	**View→Clip Work Section** /**Edit Work Section** /**New Section**
Menu	**View→Operation→Clip Work Section→Edit Work Section→New Section**
Shortcut menu	**Assembly Navigator**→right-click the **Section** folder→**New Section**

Move Object

What is it?

The new **Move Object** command enables you to quickly reposition objects in your part.

This command provides the traditional translation and rotation capabilities currently available with the **Transform** command, but also provides additional repositioning functions. You can:

- Dynamically move objects using handles.
- Move objects a specific distance along a given vector, towards a given vector, or between two points..
- Move and rotate objects along and about a specified vector.

The **Move Object** command displays a preview as the operation is being defined.

When all the objects are bodies, the **Move Object** command allows them to be repositioned without also forcing the parents to be moved. This results in the creation of a Move/Rotate feature.

Note

The Scale, Mirror, Array and Incremental Dynamics repositioning functions are only available with the **Transform** command on the **Edit** menu. This command is available with the **Essentials with full menus** and the **Advanced with full menus** roles

Where do I find it?

Toolbar	**Standard→Move Object**
Menu	**Edit→ Move Object**

Decal

What is it?

The **Decal** command has several enhancements. You can now:

- Place any number of logos on top of a textured surface.
- Better control transparency, reflectance, and placement.
- Use additional image file types (TIFF, JPEG, and PNG).
- Use **Displacement** to accentuate the differences between peaks and troughs of the displacement map.

Why should I use it?

Using this functionality makes the decal easy to create, apply, and edit.

Where do I find it?

Prerequisite	The **Rendering Style** option must be set to **Studio** on the **View** toolbar.
Toolbar	**Visualize Shape→Decal**
Menu	**View→Visualization→Decal**
Resource bar	**Materials in Part**® right click in the background® **New Entry**® **Visualization Decal**

Layer Settings

What is it?

New functionality added to the **Layers Setting** dialog box allows you to:

- Create, edit and delete layer categories, including empty categories.
- Manage layers and their categories without using the **Layer Category** dialog box.
- Control layer settings and layer and category information with a new tree list.

- Select an object to determine what layer it is on, or to quickly move or copy it to a different layer.

Why should I use it?

The ability to toggle layers and categories on and off easily, and to directly access most layer actions via the right mouse button, enables you to streamline and accelerate your workflow.

Where do I find it?

Toolbar	**Utility→Layer Settings**
Menu	**Format→ Layer Settings**

List box customer defaults

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Gateway→User Interface→** Layer Dialog Box page

Movie capture in NX

What is it?

You can now record NX interactions in an AVI movie. This is especially useful when you want to record the playback of an assembly sequence.

The movie capture commands let you start the movie recording, and pause or stop the recording when you want.

You can also specify movie settings, which apply to subsequent movies. Settings cannot be changed during the recording of a movie.

> **Note**
>
> In the **Movie Settings** dialog box, the **Compression** option lets you specify which codec to use in recording a movie. Two codecs which give very good results (small file size, and better picture quality and more frame capturing rate than with many other codecs) do not appear on Windows by default, but are available as freeware. These codecs are Microsoft MPEG-4 Video Codec V2 and Microsoft MPEG-4 Video Codec V1.

Why should I use it?

You may want to create movies for many reasons, such as the following:

- Creating customer or design review presentations.

- Documenting an NX process for training needs.
- Exchanging details of your CAD models and animations with non-NX users.

Because the NX movie capture function is integrated with other NX functions, it has the following advantages over external movie capture programs:

- The NX movie capture function can record at specified points (for example, starting at a specified step in an assembly sequence playback). It can be difficult to make an external movie program start at exactly the right time, and you could potentially lose critical data.
- Externally-captured movies are sometimes jumpy.
- If you use an external movie program, you have to jump between it and NX to control when the movie recording starts, pauses, stops, or does other operations.

Where do I find it?

Toolbar	**Movie**
Menu	**Tools® Movie**

In the assembly sequencing environment

Menu	**Tools® Export to Movie**

Text export in CGM and PDF files

What is it?

NX has greatly improved support for the export of text in CGM files, and has added support for text export in PDF files.

This improved functionality enables you to:

- Output searchable text in both CGM and PDF files.
- Use the *cgmdef.txt* file to control font substitution, if the default substitution is not acceptable.
- Map standard TrueType fonts to specific NX Fonts in the *cgmdef.txt* file.

Why should I use it?

Use this option whenever you want to have searchable text in an exported CGM or PDF files.

Note

Output to a PDF file and to a CGM file is 2D only. Therefore, only text which is flat to the screen, that is, parallel to the graphics window, is output as text. Text which is not flat is rendered as a stroked image.

Where do I find it?

Menu	**File→Export →PDF** **File→Export →CGM**
Location in dialog box	**Settings** group® **Output Text** list® **Text**

Chapter

7 Design

Modeling

Synchronous Modeling with synchronous technology

History-Free Mode

What is it?

History-Free Mode is an approach to design without linear history. Additionally, design changes are made with emphasis on modifying the current state of a model, and maintaining geometric conditions inherent in the model with synchronous relations. The history of feature operations, during geometry construction or modification, is not saved, and there is no dependence on a linear chronology of feature creation.

History-Free Mode represents an alternative to history-based modeling, where you can design rapidly in a simpler, more open-ended environment.

- You are not restricted in your model to a linear chronology of feature operations.
- Synchronous Modeling commands let you modify a model regardless of its origins, associativity, or how it was created.
- Since there is no history, there is no feature playback.
- **History–Free Mode** does not mean no features. Certain NX commands, such as **Hole**, **Blend**, **Chamfer**, and Synchronous Modeling's **Dimension** commands are treated as "local features" in this mode. You can edit them in the same way you create them. More local features will be added in future releases.

  Note

  Although there is no replay in **History-Free Mode** since the history is not kept, Synchronous Modeling features created in **History Mode** will replay like any other feature created in that mode.

Why should I use it?

Use **History-Free Mode** to quickly design and explore new concepts without having to plan modeling steps in advance. You can easily examine and question designs and ideas, test and move things around, and delete those things that do not work. **History-Free Mode** works equally well on imported or legacy models, which already do not have history.

Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling**® **History–Free Mode**
Menu	**Insert**® **Synchronous Modeling**® **History–Free Mode** **Preferences**® **Edit** tab® **Modeling Mode**® **History–Free**
Location in dialog box	**Part Navigator**® right-click **History Mode** node and choose **History–Free Mode**

Move Face

What is it?

The **Move Face** command is new, replacing the earlier command with the same name. It has the following enhancements:

- **Face Finder**: These options, available under the **Face** group, enable you to designate additional faces to move, based on how their geometry compares to that of the selected face to move.

  Many Synchronous Modeling commands now have **Face Finder** options that recognize geometric conditions between an input face and a set of scope faces within the model.

- **Transform**: This new group provides linear and angular transform methods for the selected faces to move. These methods are available in the new **Motion** list that has replaced the earlier **Type** list.

  The following new methods, are available in the **Motion** list.

  – **Distance between Points**

  – **Radial Distance**

  – **Rotate by Three Points**

  – **Distance-Angle**

  – **CSYS to CSYS**

– **Dynamic**

Note

◊ The **Dynamic** method is available only when you are in the Synchronous mode.

◊ The earlier transform methods are also available. The above methods are additional.

- Active Selection: A new Active Selection mini-toolbar for Selection Intent and **Face Finder** options now appears above the cursor position when you first select a face or curve. It lets you quickly verify and change the Selection Intent or related face rule when selecting faces and related faces. For more information, see Active Selection.

In the following example, the cylindrical faces on the model are easily moved by a specified distance and direction using the **Move Face** command.

Why should I use it?

The **Move Face** command is a useful design tool that facilitates easy design change during the design process. It is also useful in downstream applications like Tooling, Manufacturing, Simulation, where you can directly make changes to the model, regardless of feature history, and without having to send the model to the original design engineer.

Some scenarios where you can use the **Move Face** command:

- To relocate a group of faces to a different position to meet design intent.
- To change the bend angle of a sheet metal part that has no history.
- To rotate a face or set of faces about a given axis and about a point. For example, to change the angular position of a keyway slot.

Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling**® **Move Face**
Menu	**Insert**® **Synchronous Modeling**® **Move Face**
Location in dialog box	**Face to Move** group® **Face Finder** options **Transform** group® **Motion** list

Reuse commands

What is it?

Reuse commands in Synchronous Modeling let you reuse faces in a part, and if necessary change their function.

Changing the function of a part is more complex than changing its fit. You may have to take it apart, change it, and put it back together. Changing the function typically involves deleting and adding geometry.

The following new and updated commands are useful tools to use when you need to make such changes to a part.

Cut Face
Copies a face set, deletes that face set from the body, and heals the open area left in the model.

Copy Face
Copies a face set from a body, keeping the original face set intact.

Paste Face
Pastes a cut face set into a target body.

Mirror Face
Copies a face set, mirrors it about a plane, and pastes it into the part.

Pattern Face
Copies a set of faces in a circular or rectangular pattern, or mirrors them and adds them to a body.

These and other Synchronous Modeling commands are primarily suited for models composed of analytic faces (that is planes, cylinders, cones, spheres, and tori). This does not necessarily mean “simple” parts, since models with many thousands of faces are composed of these face types.

Why should I use it?

Use Synchronous Modeling **Reuse** commands to modify a model regardless of its origins, associativity, or feature history. The model could be imported, non-associative, and with no features, or it could be a native NX model complete with features.

Cut Face is useful for "suppressing" faces within a non-associative part that lacks features.

Paste Face is useful for reintroducing a set of copied or cut faces into the part.

Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling** **Insert® Synchronous Modeling® Reuse**
Menu	

Geometric transform commands

What is it?

The following new Synchronous Modeling commands let you move selected faces by transforming them based on the constraining geometry of another face:

Make Coaxial
Makes one face coaxial to another face.

Make Coplanar
Makes one face coplanar to another face.

Make Parallel
Makes one face parallel to another face.

Make Perpendicular
Makes one face perpendicular to another face.

Make Tangent
Makes one face tangent to another face.

Make Symmetric
Makes one face symmetric to another face about a plane of symmetry.

These commands work in a similar fashion to one another and share some of the same group of options, such as the following:

Motion Face Lets you select the input face whose shape you want to move or transform.

Stationary Face Lets you select the face whose geometric property is used to constrain the transforming movement of the motion face. The stationary face does not move.

Motion Group Lets you select additional faces to move in the model using the current transform command.

The **Face Guide** option within the **Motion Group** is a great new tool to designate additional faces to move in the model, based on how their geometry compares to that of the selected stationary face. It works in a way that is similar to Selection Intent, with the added benefit of collecting faces based on geometric relations.

Why should I use it?

Use these commands to modify the faces of a model regardless of its source, associativity, or feature history.

Where do I find it?

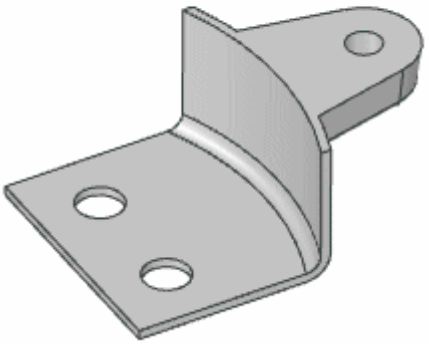
Application	Modeling
Toolbar	**Synchronous Modeling**
Menu	**Insert® Synchronous Modeling® Constrain**

Pull Face

What is it?

The new **Pull Face** command derives a volume from a face region and then modifies the model with that volume.

Although similar to the **Move Face** command, **Pull Face** adds or subtracts a new volume, while **Move Face** modifies an existing volume.



Why should I use it?

Use this new capability when you want to add or subtract a new volume on a model based on a selected face.

Where do I find it?

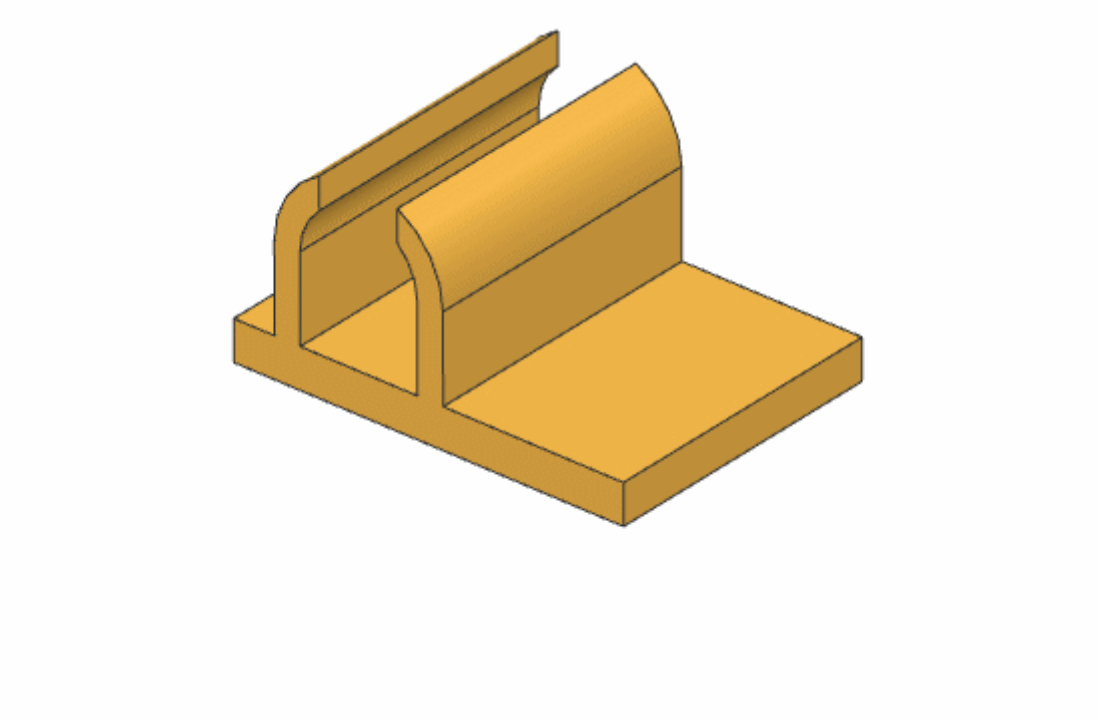
Application	Modeling
Toolbar	**Synchronous Modeling® Pull Face**
Menu	**Insert® Synchronous Modeling® Pull Face**

Linear Dimension

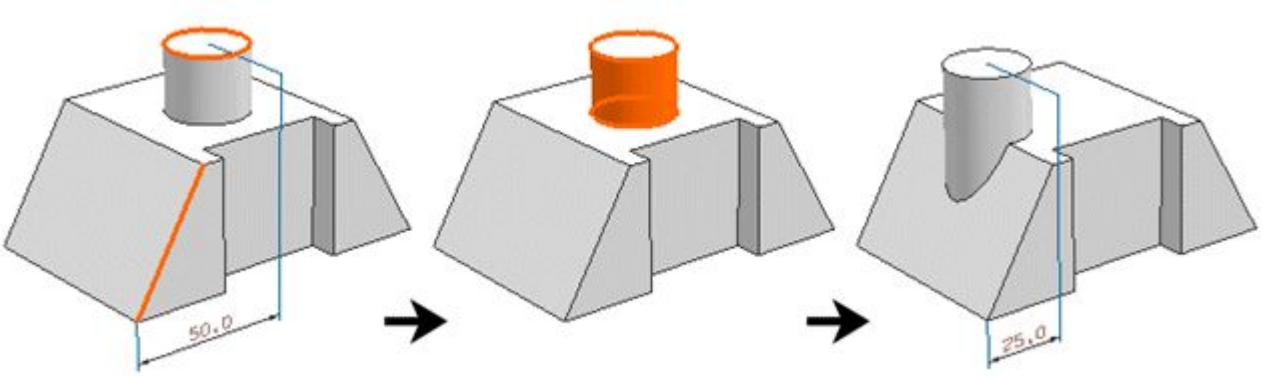
What is it?

The **Linear Dimension** command lets you move a set of faces by adding a linear dimension to a model and then changing its value.

Faces on a solid are moved in the following animation by placing a linear dimension using origin and measurement points, selecting the faces to move, and then changing the value of the dimension.



A face is moved in the next example by placing a linear dimension using origin and measurement points, selecting the face to move, and then changing the value of the dimension.



Why should I use it?

Use this command as a way to modify the faces of a model regardless of its source, associativity, or feature history.

Where do I find it?

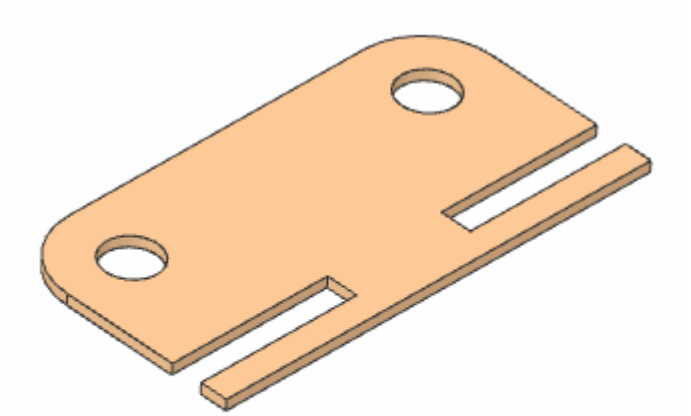
Application	Modeling
Toolbar	**Synchronous Modeling® Linear Dimension**
Menu	**Insert® Synchronous Modeling® Dimension® Linear Dimension**

Angular Dimension

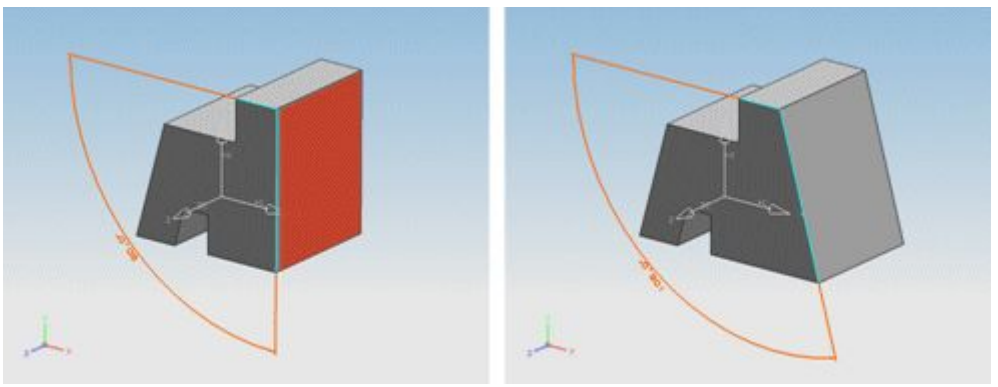
What is it?

The **Angular Dimension** command lets you move a set of faces by adding an angle dimension to a model and then changing its value.

Four faces are moved in the following animation by placing an angular dimension using origin and measurement points, selecting the faces to move, and then changing the value of the dimension.



A face is moved in the next example by placing an angular dimension using origin and measurement points, selecting the face to move, and then changing the value of the dimension.



Why should I use it?

Use this command as a way to modify the faces of a model regardless of its source, associativity, or feature history.

Where do I find it?

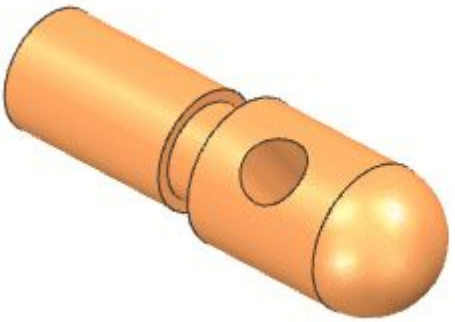
Application	Modeling
Toolbar	**Synchronous Modeling® Angular Dimension**
Menu	**Insert® Synchronous Modeling® Dimension® Angular Dimension**

Radial Dimension

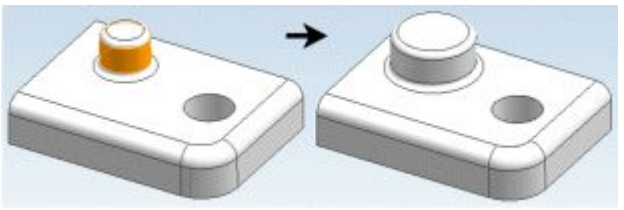
What is it?

The **Radial Dimension** command lets you move a set of cylindrical or spherical faces, or faces with a circular edge, by adding a radial dimension and then changing its value.

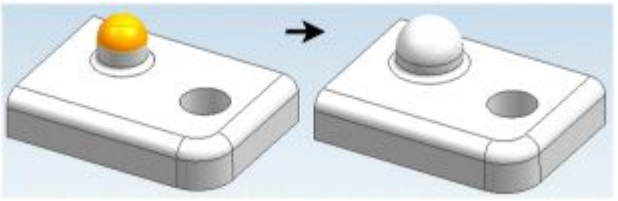
Three cylindrical faces and a sphere are moved in the following animation by placing a single radial dimension on the largest cylinder and giving the dimension a new, smaller value.



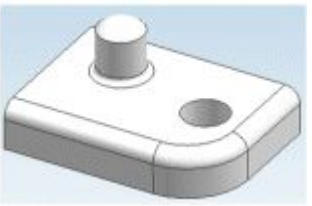
Below, a cylinder face is moved by placing a radial dimension on the highlighted face and then changing the value of the dimension.



The next example shows a sphere that is tangent to a cylinder. If you place the radial dimension on the sphere and set the **Face Guide** to recognize tangency, both the cylinder and the sphere are moved (resized) and the blend is reblended.



If tangency is not recognized, the sphere changes but does not move, and the cylinder extends to meet it.



Why should I use it?

Use this command as a way to modify the faces of a model regardless of its source, associativity, or feature history.

Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling® Radial Dimension**
Menu	**Insert® Synchronous Modeling® Dimension® Radial Dimension**

Group Face

What is it?

Use the new **Group Face** command in Synchronous Modeling to arrange faces in a group:

- The group of faces you select become a **Group Face** feature.
- Use Selection Intent to select faces for the group, or use the **Face Guide** option to select faces based on how their geometry compares to that of the selected face.
- After you create a **Group Face**, you can select its faces by selecting the feature.
- You can select the **Group Face** feature while working in other commands using the **Feature Faces** Selection Intent rule.

Where do I find it?

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

Enhanced Synchronous Modeling commands

What is it?

The following commands are enhanced.

Resize Face

New **Face Finder** options, available under the **Face** group, enable you to designate additional faces to resize, based on how their geometry compares to that of the selected face to resize.

Offset Region

New **Face Finder** options, available under the **Face** group, enable you to designate additional faces to offset, based on how their geometry compares to that of the selected face to offset.

Note

The **Adjacent Blend Face** group is no longer available. NX automatically determines any blend faces, and re-blends them.

Replace Face

- You can select more than one face as the replacement face. The selected replacement faces must be on the same body and form an edge-connected chain. If they do not meet these conditions, NX displays an error.
- You can offset the face to replace.

Note

The **Adjacent Blend Face** group is no longer available. NX automatically determines any blend faces, and re-blends them.

Where do I find it?

Application	Modeling
Toolbar	**Synchronous Modeling**

Menu	**Insert® Synchronous Modeling**

Adaptive Shell

What is it?

Adaptive Shell is a behavior of the model when using the new **History-Free Mode** (no-history), where the wall thickness is maintained when changes are made to the model using Synchronous Modeling commands.

The following new commands support Adaptive Shell:

Shell Body	Applies wall thickness and opens faces of a solid body to form a shell. This also adds the adaptive shell behavior to the body.
Shell Face	Adds faces to an existing adaptive shell or applies wall thickness to a face set.
Change Shell Thickness	Changes the wall thickness of an adaptive shell.

Why should I use it?

Adaptive Shell provides the positive aspects of a shell feature in the absence of a history.

Where do I find it?

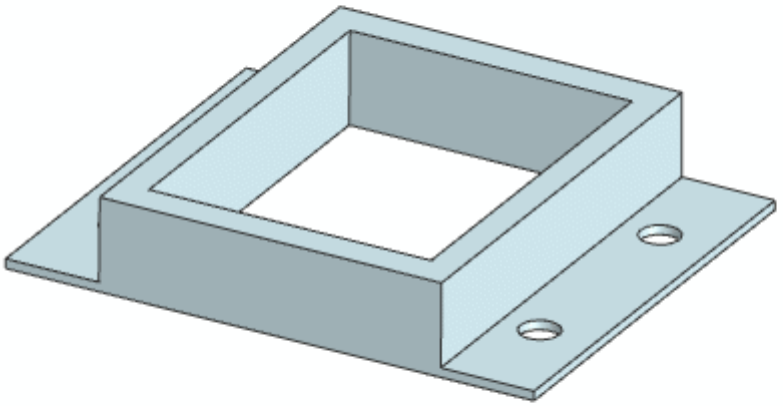
Application	Modeling
Prerequisite	Available only in the **History-Free Mode**.
Toolbar	**Synchronous Modeling**
Menu	**Insert® Synchronous Modeling® Shell**

Cross Section Edit

What is it?

Cross Section Edit is a new command that lets you modify a solid body by editing its cross section in a sketch.

You can create a cross section of an existing solid body using either a plane or a sketch on path. The Sketcher opens with a new sketch of bi-directionally constrained curves on the intersected faces of the solid body. As you modify the location and size of the sketch curves, the solid body is also modified.



A solid body is edited in a sketch by moving its cross section curves

Why should I use it?

In addition to the standard Synchronous Modeling commands, use this new command to modify a body when designing in the **History-Free Mode**.

Where do I find it?

Application	Modeling
Prerequisite	Available only in the **History-Free Mode**.
Toolbar	**Synchronous Modeling® Cross Section Edit**
Menu	**Insert® Synchronous Modeling® Cross Section Edit**

Hole

Drill Size Hole

What is it?

Drill Size Hole is a new type of hole available in the **Hole** command, that lets you create a simple Drill Size Hole feature.

You can:

- Use standards like ANSI and ISO.
- Use customized data tables to specify the options and the corresponding parameters in the **Hole** dialog box.

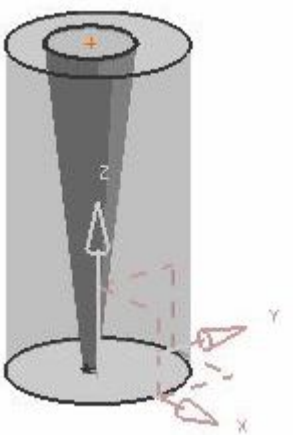
Where do I find it?

Application	Modeling
Toolbar	**Feature® Hole**
Menu	**Insert® Design Feature® Hole**
Location in dialog box	Type list

Tapered hole

What is it?

Use the **Tapered** hole form to create tapered holes. This option is available only for the **General** type of Hole feature.



Tapered hole

Where do I find it?

Application	Modeling
Toolbar	**Feature**® **Hole**
Menu	**Insert**® **Design Feature**® **Hole**
Location in dialog box	**Form and Dimensions** group® **Form** list® **Tapered**

Depth Limit options

What is it?

Three new options are added to the **Depth Limit** list. The **Until Next** option extends the hole until it reaches the selected face. This option is available for all types of Hole features.

The following **Depth Limit** options are available only for the **Screw Clearance Hole** and **Hole Series** (**End** hole) types:

- **Value** — Creates a hole of the specified depth.
- **Until Selected** — Creates a hole until the selected object.

Where do I find it?

Application	Modeling
Toolbar	**Feature Operation**® **Hole**
Menu	**Insert**® **Design Feature**® **Hole** **Screw Clearance Hole** type® **Form and Dimensions** group® **Depth Limit** list
Location in dialog box	**Hole Series** type® **Specification** group® End page® **Depth Limit** list

Edit Hole Series

What is it?

You can edit a parent Hole Series feature from a linked Hole feature that is it's child. When you select the linked Hole feature to edit, a message is displayed. You can click **Edit Hole Series** to continue editing the parent Hole Series.

Note

- The **Edit Hole Series** option is available only when you reopen the assembly that contains the Hole Series feature.
- You cannot edit the parent feature if the part is not open, or if it is not possible to make the part that contains the Hole Series feature the work part.

Where do I find it?

Application	Modeling
Toolbar	**Feature® Hole**
Shortcut menu	Right-click a linked Hole Series feature in the **Part Navigator** and click **Edit Parameters**.

Blend enhancements

What is it?

Edge Blend	Optionally capping (trimming) an edge blend has the following enhancements: • You can now select a plane as well as faces to cap an edge blend. • If you cap an edge blend with a plane, you can specify the trim location plane as also being the capping plane. • If the capping trim plane intersects the blend face in more than one location, you can specify which intersection to use to trim the blend. • If you cap the blend with one or more connected faces, they can be from the body being blended or from another body.
Face Blend	The **Multi-transition** law type (Dynamic Law) is now available with **Face Blend** when the **Radius Method** is set to **Law Controlled**.

Where do I find it?

Application	Modeling
Prerequisite	For **Edge Blend**, the **User Selected Objects** option in the **Trimming** group must be selected in order to specify your own trimming object.

Toolbar	**Feature Operation® Edge Blend** **/Face Blend**
Menu	**Insert® Detail Feature® Edge Blend/Face Blend**
Location in dialog box	**Edge Blend** dialog box® **Trimming** group® **Trim Object** and **Use Trim Plane to Cap Blend** **Face Blend** dialog box® **Blend Cross Section** group® **Radius Method® Law Controlled** and **Law Type® Multi-transition**

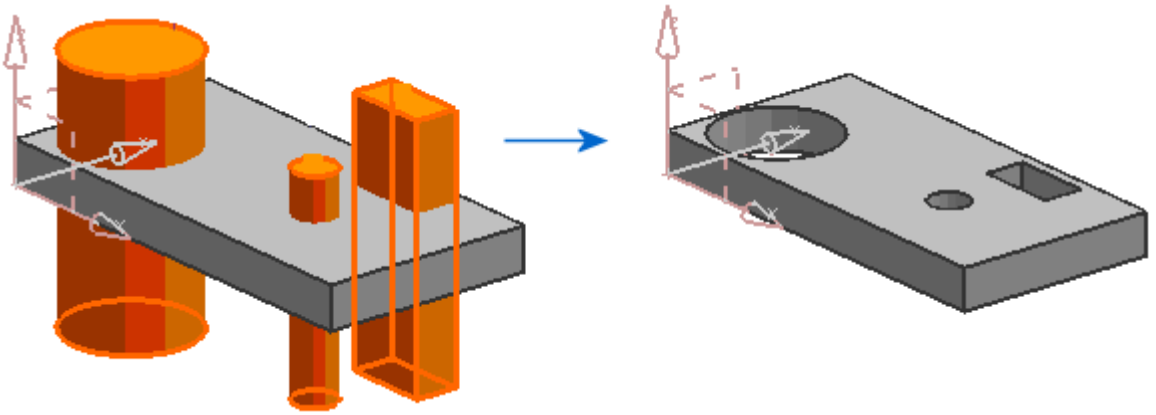
Boolean enhancements

Unite, Subtract, Intersect

What is it?

The Boolean commands **Unite**, **Subtract**, and **Intersect** now support selection of multiple bodies or a group of bodies as the tool objects.

You can also use the new **Group** option, available in the **Type Filter** list on the Selection bar, to select a group of bodies as the tool body.



Subtract command using a Group feature as the tool body

Why should I use it?

Use the enhanced Boolean commands to support multiple tool inputs.

Where do I find it?

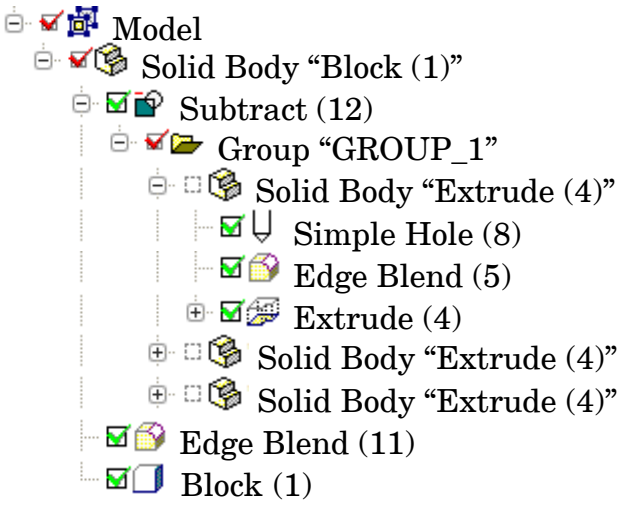
Application	Modeling
Toolbar	**Feature® Unite** **/ Subtract** **/ Intersect**
Menu	**Insert® Combine Bodies® Unite/ Subtract/ Intersect**

Boolean features in the Part Navigator

What is it?

When a Boolean operation employs multiple tool bodies, it is shown in the **Part Navigator** as a single Boolean feature.

The view of the history tree is simplified and has more pronounced branches making it easier to locate the Boolean features in the **Part Navigator**.



Part Navigator view of Boolean command [Subtract (12)] using multiple tool bodies

Where do I find it?

Application	Modeling
Toolbar	**Feature® Unite** / **Subtract** / **Intersect**
Menu	**Insert® Combine Bodies® Unite/ Subtract/ Intersect**

Split Body

What is it?

Split Body splits a solid or sheet body into multiple bodies using a set of faces or a datum plane. This command creates an associative Split Body feature that appears in the history of the model and can be updated, edited, or deleted.

Why should I use it?

This command is useful in a modeling approach where multiple parts are modeled as a single part and then split as required. For example, a housing that consists of a base and a cover can be modeled as one part and split later.

Where do I find it?

Application	Modeling
Toolbar	**Feature Operation® Split Body**
Menu	**Insert® Trim® Split Body**

Replace Feature enhancement

What is it?

Replace Feature now automatically maps the geometry of solid entities that need to be mapped.

You can then review the automatic reference resolution and change it, if required, using the new **Deviation Allowance** option in the **Replace Feature** dialog box, or change it manually.

Where do I find it?

Application	Modeling
Toolbar	**Edit Feature® Replace Feature**
Menu	**Edit® Feature® Replace**
Location in dialog box	**Mapping** group® **Deviation Allowance**

Mirror a CSYS using Instance Geometry

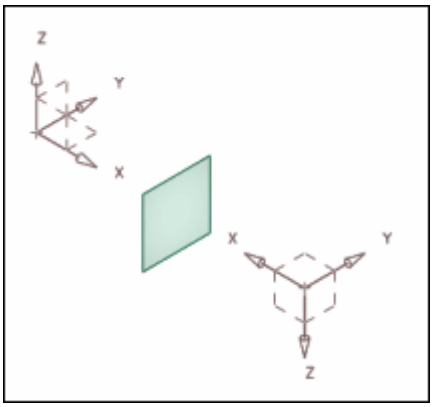
What is it?

You can now mirror both datum and non-datum coordinate systems using the **Mirror** type in **Instance Geometry**. Mirroring a coordinate system results in a right-handed CSYS. A true mirror would have resulted in a left-handed coordinate system.

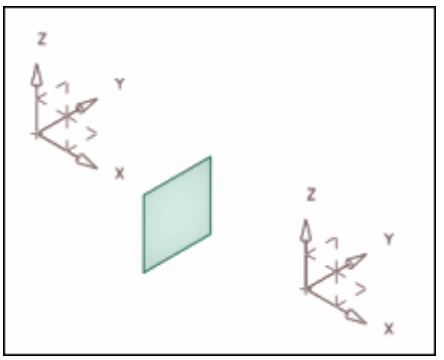
Options for the **Mirror** type include the following:

- The **CSYS Mirror Method** option lets you specify different ways to get a right-handed coordinate system: **Mirror X and Y, Derive Z**; **Mirror Y and Z, Derive X**; **Mirror Z and X, Derive Y**.

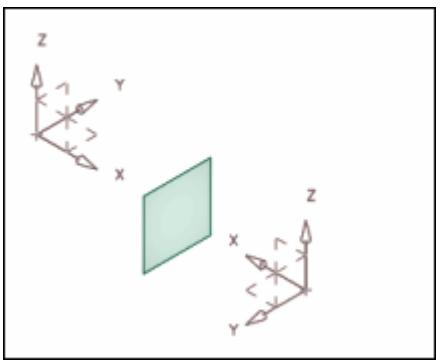
  The following figures show the results of each method when the Datum CSYS on the left is mirrored across a datum plane.



Mirror X and Y, Derive Z



Mirror Y and Z, Derive X



Mirror Z and X, Derive Y

- The new **Hide Original** option lets you hide the original geometry you have mirrored, so you can get the reflected version of an object to appear but not its original.

Why should I use it?

Getting a reflected (opposite-handed) version of an object that is both associative and editable is not available with other methods, such as **Transform** on the **Edit** menu.

Where do I find it?

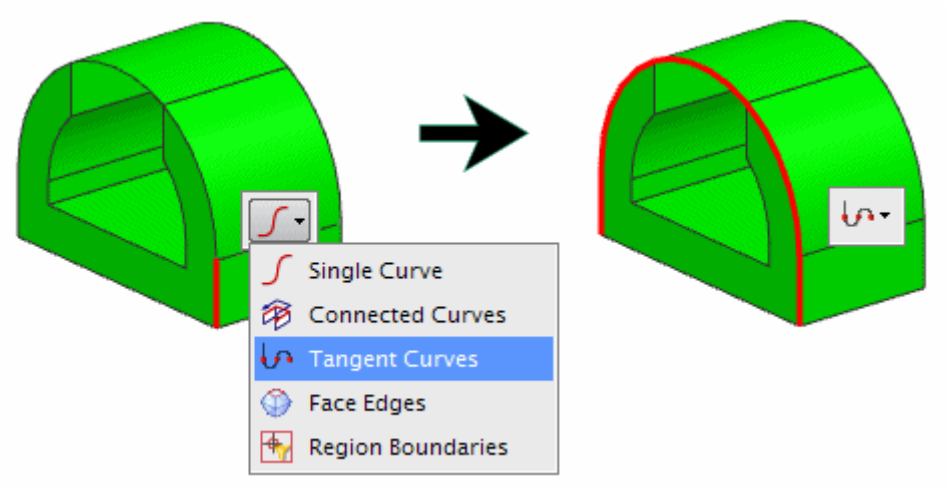
Application	Modeling
Toolbar	**Feature® Instance Geometry**
Menu	**Insert® Associative Copy® Instance Geometry**
Location in dialog box	**Type** group® **Mirror** **Settings** group® **Hide Original**

Active Selection

What is it?

An Active Selection mini-toolbar for Selection Intent and Synchronous Modeling's new **Face Finder** option now appears above the cursor position when you first select a face or curve. You can:

- Change the rule on the mini-toolbar and continue to select or deselect faces or curves.
- Dismiss the mini-toolbar by moving the cursor away from it or by clicking outside it.



Choosing Tangent Curves from the mini-toolbar to selelct a string of tangent curves

On Vista systems, the toolbar fades in and out as you move the cursor towards or away from it.

Why should I use it?

The Active Selection mini-toolbar lets you quickly verify and change the Selection Intent or **Face Finder** rule when selecting curves, faces, and related faces.

Where do I find it?

Application	Modeling
Prerequisite	The Active Selection mini-toolbar is available for Selection Intent when the rule is either **Single Curve**, **Infer Curve**, or **Single Face**. To select related faces in Synchronous Modeling, you must be in a command that has the **Face Finder** group of options, such as **Move Face**.

Feature Replay

What is it?

Use **Feature Replay** to review how features were used to construct a model.

You can:

- Manually step through the features of a model using the commands on the **Feature Replay** toolbar.
- Run, pause, and select a starting feature for an uninterrupted replay of the model using the **Automatic Feature Replay** dialog box.
- Set a time-interval for each step in a replay.
- Review features for problems during a feature replay, and fix them if necessary. The feature on which you stop the replay automatically becomes the current feature.

  Note

  Feature Replay is not a feature validation tool. Use the **Playback** command on the **Insert**® **Feature** menu for feature validation and correction.

Why should I use it?

For reviewing models, **Feature Replay** is much easier to use than the **Playback** command.

Where do I find it?

Application	Modeling
Toolbar	**Feature Replay**

	Tools® **Update**®
Menu	**Make First Feature Current** **Make Previous Feature Current** **Make Next Feature Current** **Make Last Feature Current** **Make Next Boolean Current** **Automatic Feature Replay**

Planes and datum planes using the view plane

What is it?

You can now create or specify a plane or datum plane based on the current view plane. The **View Plane** type:

- Is available for both the **Plane Constructor** and **Datum Plane** commands.
- Appears as an option for the **Specify Plane** list in supporting command dialog boxes.

Dynamic preview is available once you specify the view plane. When you rotate the view, the preview plane updates to match the new view alignment.

Where do I find it?

Toolbar	**Feature Operations**® **Datum Plane**
Menu	**Insert**® **Datum/Point**® **Datum Plane**
Location in dialog box	**Type** group® Type list® **View Plane**

Planes and datum planes using On Curve

What is it?

Planes and datum planes you specify using the **On Curve** type have the following improvements:

- You can now use Selection Intent when specifying the curve on which to create the plane or datum plane. You can, for example, select a chain of curves instead of a single curve.
- When specifying the location of the plane on curve, you can now choose a **Through Point**.

- You have new options for specifying the orientation of the plane on curve, **Normal to Vector**, **Parallel to Vector**, and **Through Axis**.

Where do I find it?

Toolbar	**Feature Operations**® **Datum Plane**
Menu	**Insert**® **Datum/Point**® **Datum Plane**
Location in dialog box	**Type** group® Type list® **On Curve**

Patch Openings

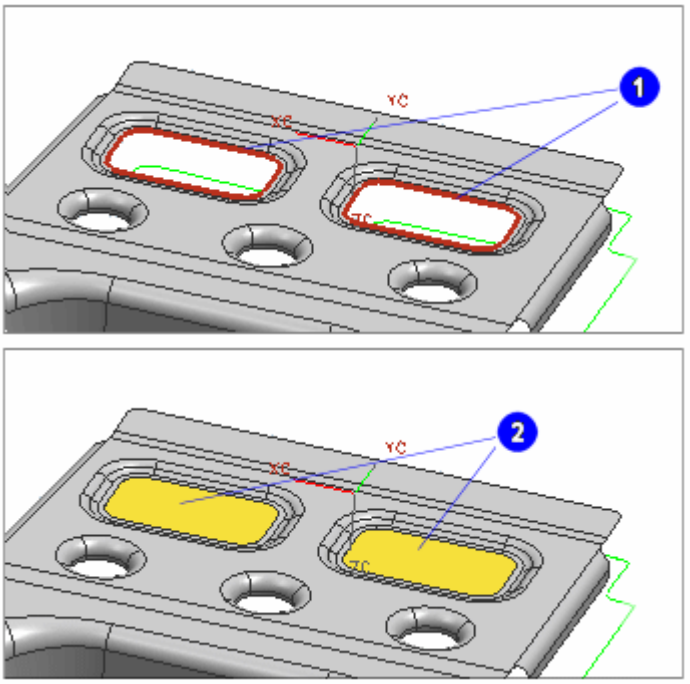
What is it?

Use the **Patch Openings** command to create sheet bodies to patch openings in a set of faces that are sewn together. These faces can be anywhere in the assembly.

When you select the faces, the software:

- Selects all openings it finds in the specified collection of faces and creates sheet bodies to patch the openings.
- Creates patches using the specified method. For example you can specify the openings to be patched using N-sided area patches, quilted patches, meshes and so on.
- Can smooth out the edges of an opening when you create face extensions along the edges.

The following graphic shows the Patch Openings feature created in a collection of sewn sheet bodies.



Patch Openings feature

1. Selected edges of openings to patch
2. Patch Openings feature

Where do I find it?

Application	Modeling
Toolbar	**Surface® Patch Openings**
Menu	**Insert® Surface® Patch Openings**

Extended basic expression data types

What is it?

Expression data types that were previously available only in Knowledge Fusion are now available in interactive NX:

Point	Defines a position by X, Y, and Z dimensions.
Vector	Defines a direction based on Cartesian I, J, and K dimensions.
Integer	Provides a numerical count without units.
Boolean	Uses a value of True or False to support alternate logical states.

Why should I use it?

These new data types provide additional parametric capability in NX.

Use the **Point** data type in commands that require the specification or reference of a position by expression. For example, you could parametrically control a **Revolve** axis location or the minimum distance location of an associative **Measure Distance**.

Use the **Vector** data type in commands that require either the input or the output (measurement) of a direction. For example, you could parametrically control an **Extrude** direction or a **Revolve** axis direction.

Use the **Integer** data type in commands that require a numerical count or quantity, such as **Instance Geometry**.

You could use the **Boolean** data type to represent the suppression status for the **Suppress by Expression** and **Component Suppression** commands.

Where do I find it?

Menu	**Tools**® **Expression**
Location in dialog box	**Expressions**® **Type**

Selection Intent in Extract

What is it?

The **Extract** command now supports Selection Intent for the **Face** type, when the **Face Option** is set to **Face Chain**.

> **Note**
>
> The **Face Option** menu list no longer appears by default for the **Face** type, because its options are now available through Selection Intent. You can restore it by selecting the **Enable legacy Face Option** menu customer default. To find this default, choose **File**® **Utilities**® **Customer Defaults**. Select **Modeling**® **Feature Parameters** and click the **Extract and Wave** tab.
>
> If you restore the **Face Option** menu list, select **Face Chain** from it to use Selection Intent face rules.

Where do I find it?

Application	Modeling
Menu	**Insert**® **Associative Copy**® **Extract**
Location in dialog box	**Extract**® **Type** group® **Type** list® **Face**

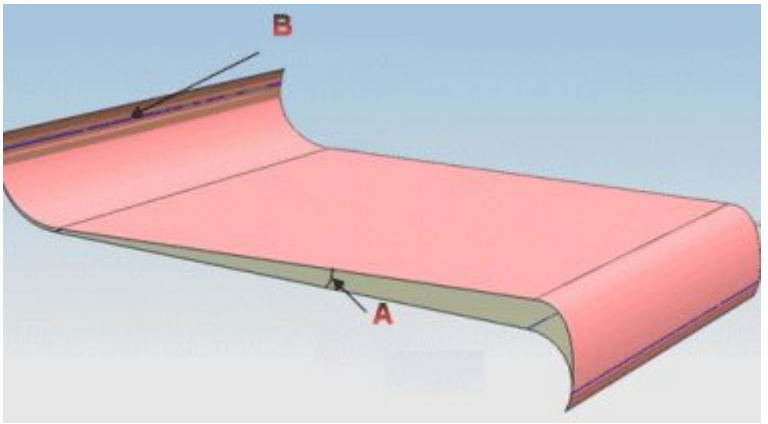
Global Shaping by Function

What is it?

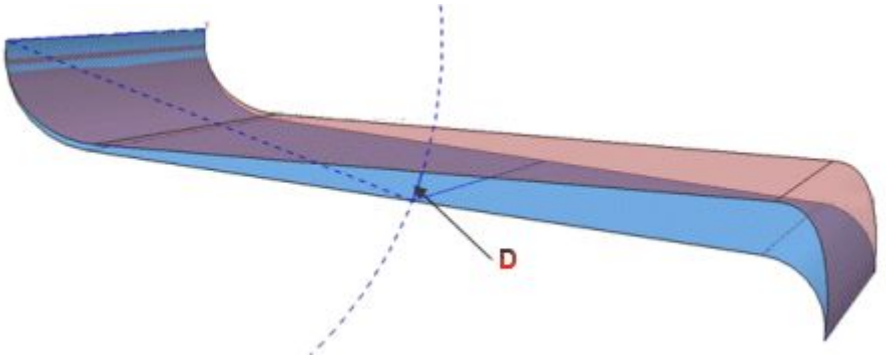
With the **Overbend** type of **Global Shaping by Function**, you can now:

- Specify region limit curves to limit the region of deformation, and region offset curves to limit the region of rotation. The region between the region offset curve and the corresponding region limit curve is smoothly blended, with no deformation at the region limit curve.

- Control rotation by specifying a chordal distance value along a curve. Previously, you could only specify an angle of rotation.



Overbend by angle input (A) with tangent continuity at bend line (B)



Overbend by distance (D) input

Why should I use it?

These improvements to the **Overbend** option give you better control over the shape of the deformation and maintain tangent continuity at the bend line.

Where do I find it?

Application	Modeling
Toolbar	**Surface® Global Shaping by Function**
Menu	**Edit® Surface® Global Shaping by Function**
Location in dialog box	**Type** group® Type list® **Overbend**

Point Set

What is it?

The **Point Set** command has the following enhancements:

- A Point Set feature now appears in the history of the model in the **Part Navigator**.
- A **Tangent Curves** option is available on the Curve Rule list on the Selection bar. This helps you select tangent curves while specifying the base geometry.

Why should I use it?

Use the **Point Set** command to easily create multiple points along a curve, along a face, at the poles of a spline or face, or at the defining points of a spline in one operation.

Where do I find it?

Application	Modeling
Menu	**Insert® Datum/Point® Point Set**

Copying faces

What is it?

You can now copy faces to paste in another location in your model or to reuse in another part file. When you use the **Copy** command to copy faces, only the faces are copied to the clipboard. The entire body is not copied.

Why should I use it?

This new capability enables you to reuse faces as opposed to recreating them. It also enables you to copy only the faces you want, instead of forcing you to copy the entire part file that contains the faces you want.

Where do I find it?

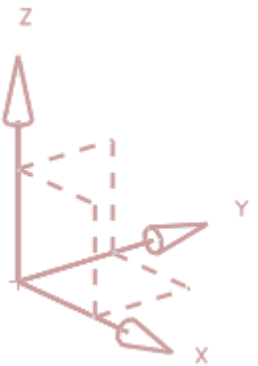
Toolbar	**Standard**® **Copy**
Menu	**Edit**® **Copy**
Shortcut menu	Right-click over a face or selected face set® **Copy**
	Part Navigator® right-click over a face feature or selected set of face features® **Copy**

Datum CSYS

What is it?

Datum CSYS has the following improvements:

- The display of a **Datum CSYS** is now better defined in the graphics window, with the axes labels clearly visible.



- **Information**® **Object** now provides expanded details for a selected **Datum CSYS**.

  Note

  See the CSYS Constructor topic for additional important enhancements that affect **Datum CSYS**.

Where do I find it?

Application	Modeling
Toolbar	**Feature→Datum CSYS**
Menu	**Insert→Datum/Point→Datum CSYS**

Block, Cylinder, Sphere, and Cone associativity

What is it?

The **Block**, **Cylinder**, **Sphere**, and **Cone** commands are now associative to their respective point, vector, and curve positioning objects.

If you subsequently move one of the positioning objects, the feature updates associatively.

Why should I use it?

This new associativity makes primitive features more useful for modeling prismatic objects such as machining fixtures.

Where do I find it?

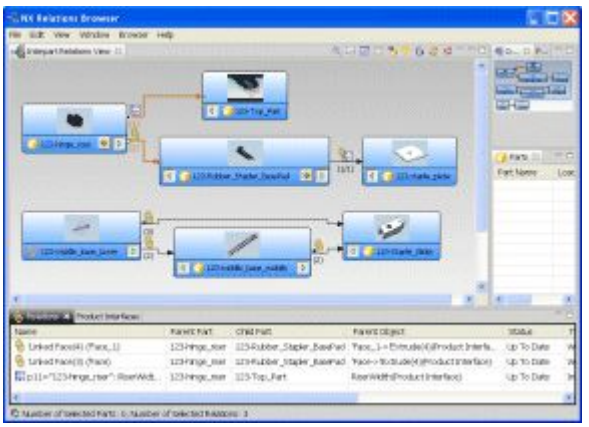
Application	Modeling
Menu	**Insert® Design Feature® Block/Cylinder/Sphere/Cone**

Assemblies

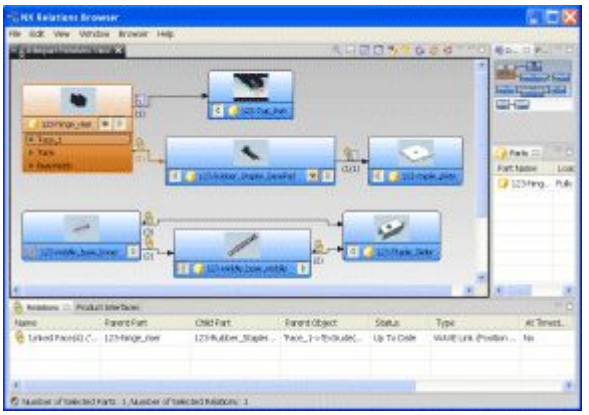
NX Relations Browser

What is it?

The new **NX Relations Browser** shows you an overall view of interpart dependency in your assembly, as well as detailed information (such as link type and status) on links and interpart expressions between parts.



If the relationship is between product interfaces, the **NX Relations Browser** shows more detailed information about which objects in the part are involved in the relationship.



In Teamcenter Integration, you can browse the relationships without loading part files.

Why should I use it?

It is easier to navigate, interrogate, and operate on dependencies, especially in complex assemblies, in a graphical-based browser than in a tree-based browser like the **Interpart Link Browser**. The **NX Relations Browser** also shows all the types of dependencies (objects, features, and parts) on a single page.

Where do I find it?

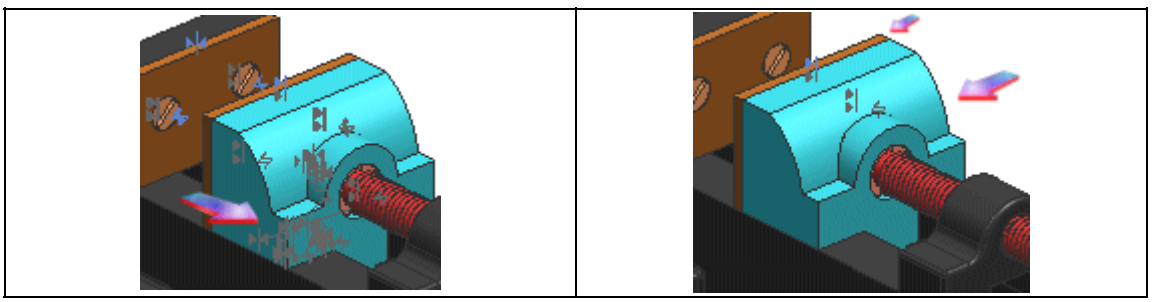
Application	Assemblies
Toolbar	**Assemblies® Relations Browser**
Menu	**Assemblies® WAVE® Relations Browser**

Assembly constraint filtering

What is it?

You can now control assembly constraint display so that only selected constraints or the constraints for a selected component are shown in the graphics window.

In the following figure on the left, the large group of constraints are so close together they are almost unreadable.



All assembly constraints	Assembly constraints for only the jaw plate and sliding jaw

Why should I use it?

When your assembly has numerous constraint markers, you cannot easily tell which ones apply to the components you are interested in. Objects you want to select or view may be hidden behind the constraint markers. Showing only the constraints you are interested in solves these problems.

Where do I find it?

Application	Assemblies
Toolbar	**Assemblies**® **Show and Hide Constraints**
Menu	**Assemblies**® **Components**® **Show and Hide Constraints**

Assembly Navigator constraint filtering

What is it?

You can now set the **Assembly Navigator** to filter out all hidden or inherited constraints.

Why should I use it?

Filtering out constraints that are hidden or inherited makes it easier for you to focus on the constraints you are interested in.

Where do I find it?

Application	Assemblies
Shortcut menu	**Assembly Navigator**® right-click in the background® **Properties**
Location in dialog box	**Filter Settings** page® **Filter Components**, then select **Filter Constraints that are Hidden** or **Filter Constraints that are Inherited** (or both)

Geometry selection for designing in an assembly context

What is it?

You can now easily select geometry outside the work part for many NX Modeling commands. You can automatically copy geometry you select outside the work part into the work part as either WAVE geometry (which updates when the source geometry changes) or non-associative geometry (which does not update).

This functionality uses two new options on the Selection bar: (1) **Select Scope** and (2) **Create Interpart Link**.

You can control interpart selection with the **Select Scope** option to select geometry from assembly components other than the work part.

Select Scope options are:

- **Entire Assembly**

  Use this option to select geometry from any component in the assembly.

- **Within Work Part and Components**

  Use this option to limit the selection of geometry to the current work part and its components, if it is a subassembly.

- **Within Work Part Only**

  Use this option to limit the selection of geometry to the current work part.

If the **Select Scope** option is not available for a command, this indicates that the command does not support this design in context functionality. In this case, you can still use one of the other traditional methods to create interpart links, such as **Wave Geometry Linker**.

When selecting geometry from a component other than the work part, you may choose to create an interpart link or to use the geometry unassociatively. To create an interpart link, turn on the **Create Interpart Link** option before making a selection. For commands where you make multiple selections, you can turn this option on or off independently for each selection.

Supported commands

Modeling functions where you can use interpart selection include the following commands:

- Sketcher
  - **Sketch**
  - **Project Curve**
  - **Project Point**
  - **Intersection Point**
  - **Intersection Curve**

- Datums
  - **Datum Plane**
  - **Datum Axis**
  - **Datum CSYS**
  - **Point**
- 3D Curve commands
  - **Basic Curves**
  - **Project**
  - **Offset**
  - **Bridge**
  - **Section**
  - **Intersect**
- Design Features commands
  - **Extrude**
  - **Revolve**
  - **Block**
  - **Cylinder**
  - **Cone**
  - **Sphere**
- Associative Copy commands
  - **WAVE Geometry Linker**
  - **Instance Geometry**
- Combine Bodies and Trim commands
  - **Unite**
  - **Subtract**
  - **Intersect**

  - **Assembly Cut**
  - **Trim Body**
- Surface creation commands
  - **Four Point Surface**
  - **Ruled**
  - **N-Sided Surface**
  - **Through Curves**
  - **Through Curve Mesh**
  - **Studio Surface**
  - **Swept**
  - **Variational Sweep**

    Note

    Not every selection option of each command is capable of creating an interpart link. The **Create Interpart Link** button will be active when it is possible to create an associative link for the selection.

The **Allow Associative Interpart Modeling** customer default lets you specify one of the following behaviors:

- **Yes**

  You can allow interpart links when you work in supported Modeling commands.

- **Yes, but not within commands**

  You can specify that links be created, but only through the **WAVE Geometry Linker** command.

- **No**

  You can disallow any interpart linking.

Where do I find it?

Select Scope and **Create Interpart Link** options:

Application	Modeling, Assemblies
Graphics window	Selection bar

Allow Associative Interpart Modeling Customer Defaults option:

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Assemblies→General**® Interpart Modeling page

Facet selection

What is it?

You can now select nonassociative facet geometry when you select an object for the following dialog boxes:

- Vector Constructor
- Point Constructor
- Plane Constructor

You can also select facet geometry when you specify a vector, point, plane, or orientation in the following dialog boxes:

- **Insert Motion** (in the assembly sequencing environment)
- **Extraction Path** (in the assembly sequencing environment)
- **Move Component** when **Type** is set to any option except **Translate**. Also, when **Type** is set to **Dynamic**, the drag handles must be in the origin or direction handle orientation.
- **Measure Distance** when **Type** is set to **Distance** or **Projected Distance**
- **Measure Angle** when **Reference Angle** is set to **Vector**
- Dynamic **Sectioning** when specifying a plane

Why should I use it?

Making faceted representations available for more functions lets you use the advantages of lightweight loading (such as faster performance) more often.

Deform Components

What is it?

The **Deform Component** function is enhanced in the following ways:

- You can use the new **Update** option on the **Deform Component** dialog box to update deformed components.
- Deformed parts can have more than one body.

Why should I use it?

You no longer need to delete and re-create outdated deformed components. The **Update** option does that for you.

It is useful to have deformed components support multiple bodies in cases where:

- An assembly contains more than one instance of the deformed component, and you want to show different instances in different forms (for example, you want to show a detailed spring for some instances, and a simplified representation for others).
- The deformable part contains multiple bodies that must be deformed together.

Where do I find it?

Application	Assemblies
Prerequisite	The part must be defined as a deformable part. See the *Assemblies* Help for more information.
Menu	**Assemblies® Components® Deform Part**

WAVE General Relinker

What is it?

The **General Relinker** dialog box now has the following enhancements and changes:

- You can specify whether suppressed components should be included as source objects by selecting or clearing the **Include Suppressed Components** check box in the **Settings** group.
- You can define the source searching scope for the relinking operation by setting the new **Source Scope** option (in the **Relink** group) to either **Parts in Assembly** or **Parts in Session**.
- You can control whether the **General Relinker** looks for non-broken WAVE links, as well as broken ones, with the new **Include Non-broken WAVE Links** check box. When this check box is not selected, update performance is faster, and non-broken links that share the same naming convention as broken links are not reparented.
- **Relink Scope** (in the **Relink** group) is now **Target Scope**.
- **Face/Curve Direction Adjustment**, in the **Settings** group, is now called **Face Normal/Curve Direction Adjustment**.

The following functions are now supported in the **General Relinker**:

- **Delay Update**
- Linked CSYS
- Linked composite curves

The **General Relinker** now supports interpart expressions with one or more spaces in the component part file names.

Where do I find it?

Menu	**Assemblies**® **WAVE**® **General Relinker**

Replacement Assistant

What is it?

The **Replacement Assistant** dialog box now provides the following:

- Automatic matching techniques (matching of objects by name, inferring from accepted matches, and geometric matching).
- Automatic progression through unmatched objects when you are manually matching objects.

Also, you can now:

- Use face and curve rules to collect faces and edges for matching.
- Perform multiple edit operations such as **Accept** and **Delete**.

Why should I use it?

These enhancements save you time on manual remapping, especially in large assemblies or when the new version has topology changes or deletions.

Where do I find it?

Application	Modeling
Prerequisite	You must be in the editing version of the **WAVE Geometry Linker** or **Extract** dialog boxes for appropriate linked or extracted geometry. See the *Assemblies* Help for more information about the **Replacement Assistant**.
Shortcut menu	**Part Navigator**® right-click a linked or extracted feature node® **Edit Parameters** or **Edit with Rollback**

Move Component

What is it?

You can now create one or more copies of components in the **Move Component** dialog box either automatically or manually.

Copies are created automatically when you move a component by any of the following methods:

- Translation, when you enter a linear handle value
- Rotation, when you enter a rotation handle value
- Snapping the origin, linear, or rotation handles
- Dragging the handles
- Using any of the **Type** options in the **Move Component** dialog box except **By Constraints**

  Note

  You can create intermediate copies when you use any of these methods except dragging the handles.

If you want more control over when a copy is made, you may want to use the **Manual Copy** mode. In this mode, a copy is created when you choose **Create Copy** either in the **Move Component** dialog box or on the handles shortcut menu. After you create a manual copy and move it into position, you can choose the **Repeat Transformation** options to quickly create and position more copies.

Why should I use it?

You may find the ability to create copies of a component at specific points in a motion useful when, for example, you want to show limits or create a simple motion path to perform analysis operations.

Where do I find it?

Application	Assemblies
Toolbar	**Assemblies® Move Component**
Menu	**Assemblies® Components® Move Component**
Location in dialog box	**Copy® Mode**

Replace Component

What is it?

The multiple **Substitute Component** dialog boxes are now combined into a single **Replace Component** dialog box.

The **Open As** and **Open Assembly As** functions that were available in the **Assembly Navigator** are also now consolidated into **Replace Component**, as follows:

- If the **Maintain Relations** check box is not selected, the selected components are removed, and the replacement part is added as a new component.
- If the **Maintain Relations** check box is selected, NX maintains as many relations as possible using the open-as capabilities. You may need to save modified parts, because the selected components are closed and then reopened as the replacement part.

Why should I use it?

You need less time to replace a component.

Where do I find it?

Application	Assemblies
Toolbar	**Assemblies® Replace Component**
Menu	**Assemblies® Components® Replace Component**

Selection intent in WAVE Geometry Linker

What is it?

Improvements to the **Extract** function in the **Modeling** application also apply to the **WAVE Geometry Linker**.

Click here for more information:

Selection Intent in Extract

Where do I find it?

Toolbar	**Assemblies® WAVE Geometry Linker**
Menu	**Insert® Associative Copy® WAVE Geometry Linker**
Location in dialog box	**Type® Face**

Assembly Navigator

What is it?

The **Assembly Navigator** has the following enhancements, which appear in both native NX and Teamcenter Integration for NX. See the Teamcenter Integration for NX section for more information and additional changes:

- Non-geometric components (NGCs) are now available in NX assemblies and can be displayed in the **Assembly Navigator**. An example of an NGC is glue, which has no geometry to display in the graphics window.

- You can specify the quantity or change the NGC status of a component on the Assembly page of the **Component Properties** dialog box.

- You can now update a parts list by updating the assembly structure in the **Assembly Navigator**.

- A new **Info** column shows additional information about a component when that information does not change the default behavior of the component, such as the "Linked part not work part" property. If the component has only one property appropriate for the **Info** column, the column uses a specific icon for that property; for example, a referenced component has a reference book icon. If the component has more than one property, the **Info** column shows a generic information icon, and details are provided in the icon's tooltip.

Restore last NX session

What is it?

You can now use bookmarks to save and restore the following characteristics of your NX session:

- The load status of parts

- The configuration rules

- The Assembly Navigator's expand/collapse state

The enhanced bookmarks also support sessions that have assemblies with both transient and persistent nodes.

Bookmarks are stored in the native NX file system, regardless of whether you are in native NX or Teamcenter Integration for NX.

Why should I use it?

This function lets you save time by preserving the state of your current NX session, so you need not reset the above characteristics when you return to your work in later sessions.

Where do I find it?

To save a bookmark manually

Menu	**File**® **Save Bookmark**

To save a bookmark automatically when you close your session

Prerequisite	You must restart your NX session after setting this customer default.
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Gateway**® **General**® **Part** tab® **Write Bookmark File on Exit**

To open a new session using a previously-saved bookmark

	In native NX
	File® **Open**
	In Teamcenter Integration for NX
Menu	**File**® **Open Bookmark**

Reference sets

What is it?

When you create a new reference set, the new **Add Components Automatically** check box lets you specify whether you want new components to be automatically added to the reference set.

Note

This capability existed before NX 6.0, but it appeared in a separate dialog box.

Also, you can control the initial setting of the **Add Components Automatically** check box for user-defined reference sets with a customer default.

If you create a reference set while the **Add Components Automatically** check box is not selected, new components you subsequently create are excluded from the reference set. If you then reopen the **Reference Sets** dialog box and

select the **Add Components Automatically** check box, you receive a message asking if you want to add the excluded components to your reference set.

The **Information** window now shows whether your reference set has excluded components.

Where do I find it?

Menu	**Format**® **Reference Sets**
Location in dialog box	**Settings**® **Add Components Automatically**

Customer default for user-defined reference sets

Menu	**File**® **Utilities**® **Customer Defaults** **Assemblies**® **Site Standards**® Reference Sets
Location in dialog box	page® **User Reference Sets**® **Add Components Automatically**

Load controls for multi-CAD JT files

What is it?

The controls for selective loading of component data from multi-CAD JT datasets in Teamcenter Integration are more in line with those for loading NX data. The setting of the **Use Partial Loading** assembly load option now has the following behavior:

- If **Use Partial Loading** is not selected, all data is loaded from the JT dataset
- If **Use Partial Loading** is selected, only the information required by your current reference set is loaded

This is similar to the partial loading behavior for NX files, where geometry loading is controlled by the current reference set when the **Use Partial Loading** assembly load option is selected.

You can set a customer default to make an exception for the automatic loading of JT Brep data from JT files.

The new **Multi-CAD** column in the **Assembly Navigator** shows which assembly components are loaded from JT files.

Why should I use it?

You now see fewer differences between the handling of multi-CAD JT data and the handling of data created in NX, which makes the use of multi-CAD data more seamless.

Because Breps stored in a JT file using the JT Brep format can take longer to load than Breps stored in the XT Brep format, you may sometimes want to prevent JT Brep data from loading automatically.

Where do I find it?

Menu	**File® Options® Assembly Load Options**
Location in dialog box	**Scope® Use Partial Loading**

The customer default to control whether JT Brep data is loaded when other data is loaded

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Gateway® JT Files® Extract Exact Data® Initially Extract Exact Data**

Sketcher

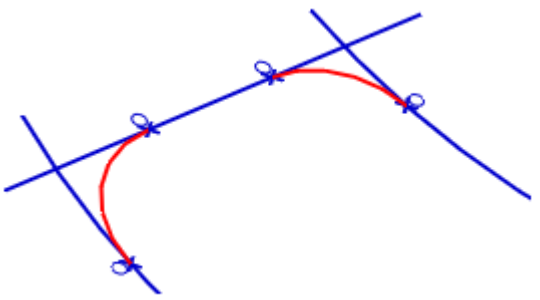
Trim Recipe Curve

What is it?

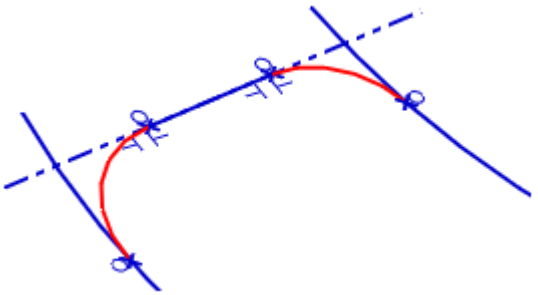
Use this command to associatively trim curves that you project or intersect into a sketch. The curves you project or intersect into a sketch are called the *Recipe Chain*. NX:

- Creates a **Trim** constraint (⊣⊢) where the bounding objects intersect the trim curves.
- Updates the Recipe Chain when you edit the parent curves.

The blue curves are projected to a sketch. The red arcs are sketch curves that serve as boundary objects for the trim.



Trimmed portions of the Recipe Chain become reference curves (- - -).



Where do I find it?

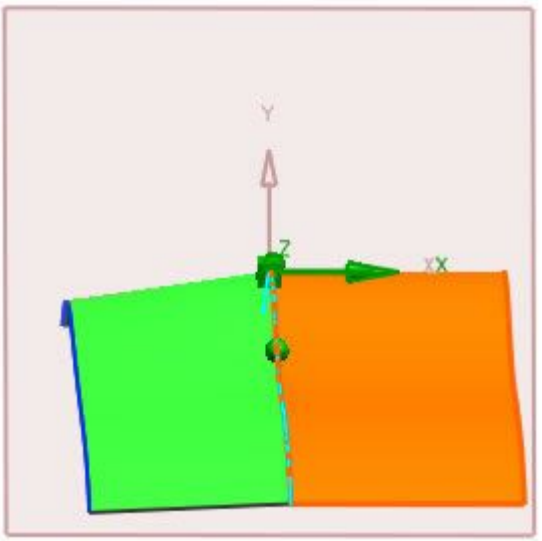
Task environment	Sketch
Prerequisite	You must create a Recipe Chain in a sketch using either the **Project Curve** or **Intersection Curve** command.
Toolbar	**Sketch Tools→Trim Recipe Curve**
Menu	**Edit→Curve→Trim Recipe Curve**

Orient Sketch on Path axes to a face or a curve

What is it?

Enhancements to **Sketch on Path** let you control the orientation of the sketch axes using these methods:

Relative to Face	Ensures that NX orients the sketch to a face, either inferred or explicitly selected. The path location you select determines the direction of the sketch plane normal.
Use Curve Parameters	Ensures that NX orients the sketch using curve parameters, even if the path is an edge, or is part of a feature that lies on a face.
Automatic	Preserves the default **Sketch on Path** orientation behavior from NX 5. That is, if you select a curve, NX orients sketch axes using curve parameters. If you select an edge, NX orients the sketch axes relative to the face, or one of the faces, that owns the edge.



Sketch on Path with axes relative to face on the right

Why should I use it?

Use the new methods to specify your design intent explicitly, especially when you orient a **Sketch on Path** for the **Variational Sweep** command.

Where do I find it?

Task environment	Sketch
Toolbar	**Feature→Sketch**
Menu	**Insert→Sketch**
Location in dialog box	**Create Sketch→Sketch Orientation→Method**

Sketch Style, Parameters, and Preferences Changes

What is it?

The Sketch task environment has the following new and modified dialog boxes:

Annotation Style	Sketcher now uses the same **Annotation Style** dialog box as Drafting for formatting individual dimensions.
Sketch Parameters	This new dialog box gives you easy access to all the dimensions in the current sketch. Edit a dimension by typing in a value or moving a slider.

Sketch Style	This new dialog box lets you control settings for the active sketch, including the display of sketch dimension labels, as well as settings for inferred constraints, fixed text height, and object color display.
Sketch Preferences	The **Sketch Preferences** dialog box now controls style settings for future sketches, as well as settings for the current NX session and the current part. To change style settings for the active sketch, use the new **Sketch Style** dialog box.

Where do I find it?

Task environment	Sketch
Menu	**Sketch→Sketch Style** **Edit→Sketch Parameters** **Edit→Style** **Preferences→Sketch**
Shortcut menu	Right-click a dimension →**Style**.

Shape Studio

Best Fit Alignment

What is it?

With the new **Best Fit Alignment** command you can transform and align geometric objects to each other in an iterative process using a true best fit method.

The command consists of the following:

- **Mobile Object**. These are the objects to be transformed and can be of almost any object type.
- **Source Object**. These objects determine where the transformation originates. They can be facet bodies, curves, surface bodies, sheet bodies, or solid bodies. In most cases, the source object is also one of the selected mobile objects.
- **Destination Object**. These objects determine where the transformed objects are placed. They can be facet bodies, curves, surface bodies, sheet bodies, or solid bodies, depending on the type of the source object.

- You can specify fitting constraints to control the degree of freedom of movement of the transformation, or you can use a global pre-alignment orientation.

  Note

  The former **Best Fit Alignment** command is still available, but is now called **Multi-Patch Alignment**. It is intended for a specific alignment task, that of aligning multiple facet bodies to each other (for example, multiple or overlapping optical scans from different views of the same object).

Why should I use it?

You can align two slightly different geometric shapes so they match as closely as possible. You would typically use this command to align a facet body to a solid body or sheet body, but a variety of other combinations is also supported.

Where do I find it?

Application	Modeling
Menu	**Edit® Facet Body® Alignment® Best Fit**

Fill Hole

What is it?

Fill Hole lets you quickly close holes in facet bodies. By applying a smoothness type, you can correctly close holes in flat areas as well as in areas with high curvature.

You can:

- Remove holes from an imported facet body.
- Fill a selected hole with triangles using the **Fill Hole** type.
- Fill holes that have isolated islands of triangles within them using the **Fill Island** type.
- Connect two selected open edges with a bridge of triangles using the **Bridge Gap** type.

Why should I use it?

Closed, water tight facet bodies are required for simulation and rapid surfacing.

Where do I find it?

Application	Modeling
Menu	**Edit® Facet Body® Fill Hole**

Rapid Surfacing

What is it?

The **Rapid Surfacing** command has been enhanced to help you reverse engineer a surface model from a facet body. The following options have been added:

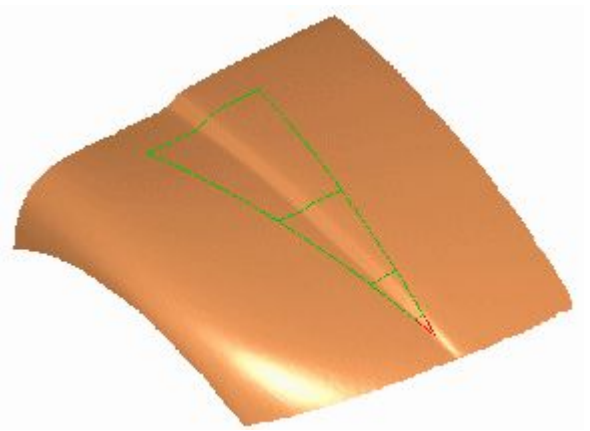
Draw on Facet Body Boundary	Creates mesh curves on the boundary of a facet body.
Subdivide Loop	Adds new patches to a valid loop of mesh curves.
Delete Mesh Nodes	Deletes all mesh curves connected to the mesh node.
Drag Curve Point	Edits a mesh curve by dragging a curve point on the facet body.
Drag Mesh Node	Edits a curve mesh by dragging a node on the facet body.

You can now:

- Set up a curve mesh structure with T-junctions.
- Undo an operation during creation or modification of the curve mesh network.

There is now support to set up a curve mesh structure with T-junctions.

You can now undo an operation during creation or modification of the curve mesh network.



Draw on Facet Body Boundary and Subdivide Loop

Why should I use it?

Use this command to quickly develop a surface on facet bodies. You can draw patches on facet bodies and edit and subdivide them to create a high patch density curve mesh network in detail areas, without increasing the patch density in areas where it is not required.

Where do I find it?

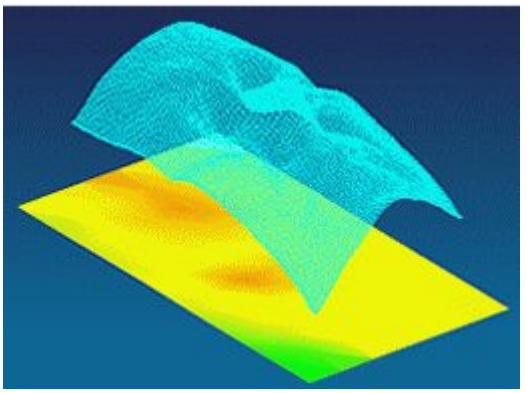
Application	Shape Studio
Menu	**Insert→Surface→Rapid Surfacing**

Deviation Gauge

What is it?

Deviation Gauge now allows you to:

- Check for deviation between facet bodies.
- Perform the deviation involving trimmed surfaces more accurately.



Facet bodies as target and resulting color map of deviation

Why should I use it?

Use this command to display deviation information (such as maximum or minimum deviations or color map) between target facet bodies and one or more reference objects.

Where do I find it?

Application	Modeling / Shape Studio
Toolbar	**Analyze Shape®** (Shape Studio)
Menu	**Analysis® Deviation® Gauge**

Drafting

View dialog box enhancements

What is it?

The following view dialog boxes have been enhanced to meet the new look and feel of the Drafting NX 6 user interface:

- Drawing
- Base
- Projected
- Detail

Annotation Leader and Origin dialog box enhancements

What is it?

The following are a few examples of dialog boxes that have embedded origin and leader blocks that enable you to add, remove, or edit leaders from annotations in Drafting and PMI:

- **Note**
- **Weld Symbol**
- **Identification Symbol**
- **Datum Feature Symbol**
- **Feature Control Frame**
- **PMI Notes dialogs**

Why should I use it?

These common dialog groups allow you to easily specify and edit annotation origins and leaders. These groups now support handles and on screen dialogs for added ease of use. See the leader handle section for additional information.

Where do I find it?

Application	Drafting

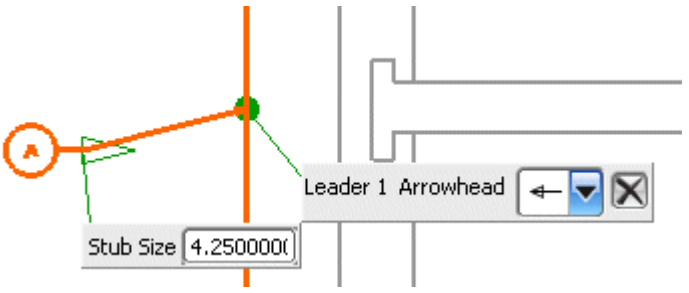
Toolbar	**Annotation® Note**
	Annotation® Weld Symbol
	Annotation® ID Symbol
	Annotation® Datum Feature
	Annotation® Datum Target
	Annotation® Feature Control Frame
Menu	**Insert® Note**
	Insert® Symbol® Weld Symbol
	Insert® Symbol® Identification Symbol
	Insert® Datum Feature
	Insert® Datum Target
	Insert® Feature Control Frame

Leader on screen interaction

What is it?

The annotation leader handle allows you to:

- Reassociate the leader to a different terminating object
- Change the stub size
- Change the leader arrowhead type



Leader handle

Why should I use it?

Use the leader handle to reassociate the leader, change the arrowhead type during creation, or edit without opening any additional dialogs.

Where do I find it?

Application	Everywhere the Annotation Leader Block is used.
Menu	**Insert**® **Symbol**® any symbol type

Sketch in Drafting

What is it?

You can now create and edit sketches directly in drafting member views or on the drawing sheet without invoking the Sketcher Task Environment. Sketches are drawn in the active member or drawing view, which you activate from the part navigator, on the sketcher tools toolbar, or via the mouse buttons.

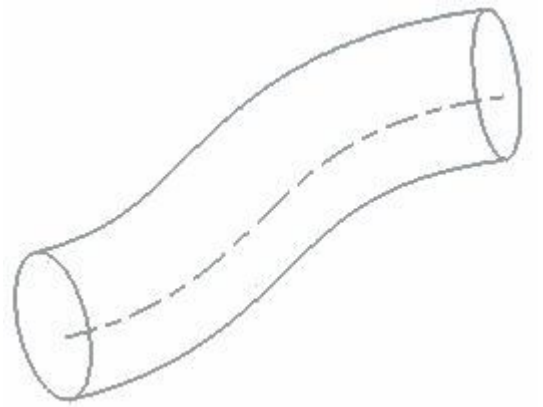
Where do I find it?

Application	Drafting
Toolbars	• **Sketch Tools**

2D and 3D Centerlines

What is it?

You can create centerlines on tubes, bends, swept faces, and between arbitrary spline curves when you use the 2D and 3D centerline commands. The 2D centerline function lets you select two arbitrary curves to create a centerline. The 3D centerline function lets you select a face such as a tube, bend, or swept face to create a centerline.



Tube with centerline

Why should I use it?

This provides an easy and direct way of creating centerlines for arbitrary geometry.

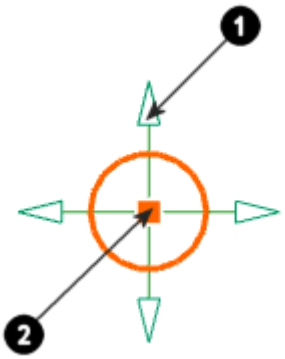
Where do I find it?

Application	Drafting
Toolbar	**Centerline® 2D Centerline** and **3D Centerline**
Menu	**Insert® Centerline® 2D Centerline** and **3D Centerline**

Centerline handles

What is it?

You can use centerline handles to modify associativity objects and lengths of centerline segments. Centerline handles provide on-screen interaction to lengthen or shorten centerlines. The handles appear when you create or edit a centerline symbol. Handles are not supported for the Automatic Centerline command.



1 — Drag handle to size centerlines

2 — Associativity Handle

Where do I find it?

Application	Drafting
Toolbar	**Centerline®** any centerline command
Menu	**Insert® Centerline®** any centerline command

Store custom symbol in part

What is it?

A new library has been added to the **Create Custom Symbol** and the **Custom Symbol** commands. This library stores custom symbols within the part rather than in the operating system in order to improve the process of defining and placing custom symbols.

Why should I use it?

You can use the new functionality when you need to define a symbol that is only going to be used in the current work part.

Where do I find it?

Application	Drafting
Toolbar	**Annotation® Custom Symbol**
Menu	**File® Utilities® Create Custom Symbol** **Insert® Symbol® Custom Symbol**.
Location in dialog box	**Libraries** group® **Part**

Smash Custom Symbol

What is it?

Smash Custom Symbol lets you break a custom symbol into its constituent pieces.

Why should I use it?

Use this feature when you need to reference or edit the individual pieces of the symbol. For instance, you may want to dimension to an object within the symbol.

Where do I find it?

Application	Drafting
Toolbar	**Annotation® Smash Custom Symbol**
Menu	**Edit→Smash Custom Symbol**.
Location in dialog box	**Libraries** group® **Part**

Feature Control Frame

What is it?

When you create Feature Control Frame (FCF) symbol annotations, the following additional tolerance modifiers are available:

- Free State modifier
- Projected modifier and Projected value
- Circle U modifier and Circle U value
- Statistical Tolerance modifier

For each FCF symbol annotation, you can create:

- A unit basis value
- A square zone tolerance modifier
- An FCF with a maximum tolerance
- An FCF with a datum reference with a Free State modifier
- Compound datum references. Each datum reference in a compound datum reference can have its own Material Modifier Conditions and Free State modifier.

FCF with Unit Basis Value, compound datum references, and Free State symbols

Why should I use it?

You can now define a Feature Control Frame more precisely.

Where do I find it?

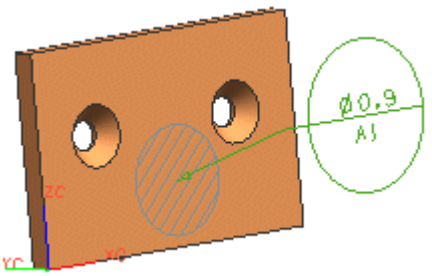
Application	Drafting and PMI
Toolbar	(Drafting) **Annotation® Feature Control Frame**
Menu	(PMI) **PMI Datum and FCF® Feature Control Frame** (Drafting) **Insert® Feature Control Frame** (PMI) **PMI→Feature Control Frame**

New datum target types

What is it?

New types of datum targets that you can create for Drafting and PMI are:

- Rectangular
- Circular
- Annular
- Cylindrical
- Arbitrary



Circular datum target example

For PMI rectangular, circular, annular, and cylindrical datum targets, you can select an existing PMI region to inherit its size and shape. Otherwise (for PMI datum targets with no selected PMI region, and for all Drafting datum targets), the datum target's parameters are determined by options in the **Datum Target** dialog box.

You can specify whether the datum target leader has an X symbol where it terminates to its target, as you can for the existing point and line datum targets.

Where do I find it?

Application	Drafting and PMI
Toolbar	(Drafting) **Annotation**® **Datum Target** (PMI) **PMI**® Datum and FCF ® **Datum Target** (Drafting) **Insert**® **Datum Target**
Menu	(PMI) **PMI**® **Datum Target**
Location in dialog box	**Type**

Crosshatch

What is it?

Crosshatch now allows you to:

- Create and edit crosshatch using the same dialog.
- Explicitly choose the method for defining the crosshatch boundary to either select curves or specify a screen position for automatic boundary.
- When selecting curves to define a crosshatch boundary the Selection Intent rules are used to determine what curves get automatically chained.
- Add, remove, and edit multiple boundaries in a single crosshatch entity via the set list.
- Access the crosshatch parameters on the same dialog as create and edit.

Why should I use it?

You can now create and edit crosshatch via a new dialog box that:

- Lets you select a boundary by curves or clicking inside the boundary of closed curves.
- Shows boundary curves in a list box.
- Lets you select annotation to exclude.
- Lets you select a crosshatch file, pattern, distance, angle, color, width and tolerance.

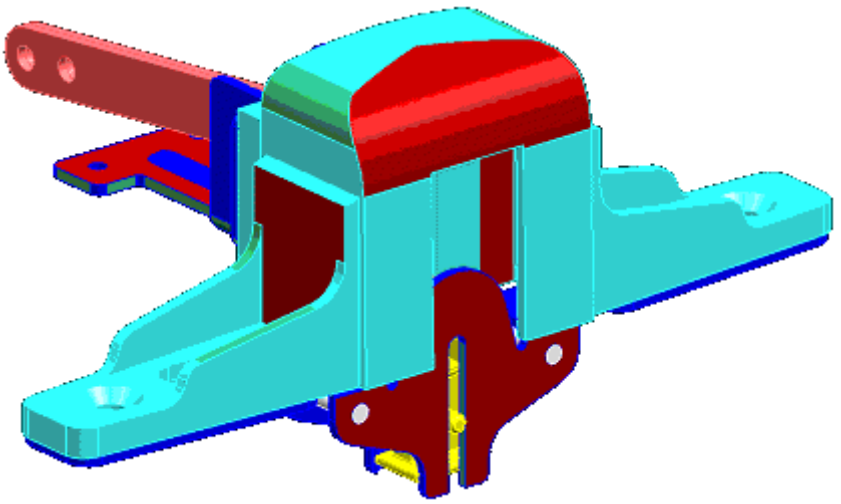
Where do I find it?

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

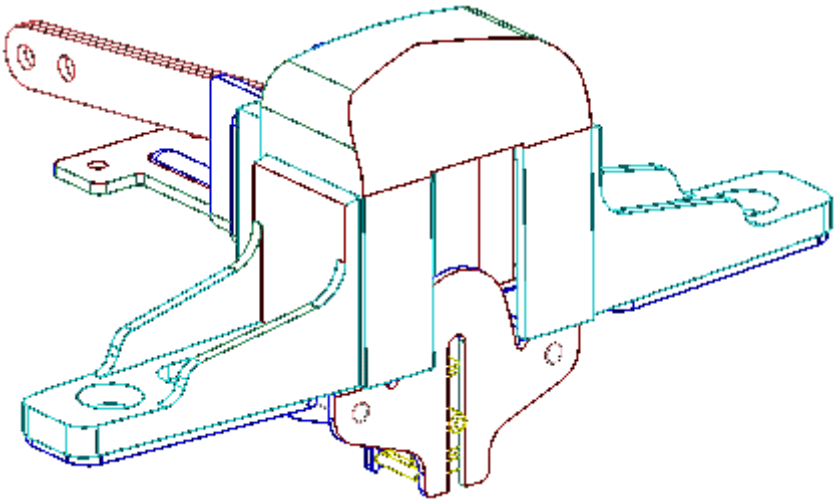
Wireframe color from face

What is it?

You can display the edges and drafting curves of member views in the color of the associated face. The color assigned to an edge or drafting curve can be overridden by Hidden Line Color, Visible Line Color, and View Dependent Edits. Previously, wireframe and drafting curves were always displayed in the color of the associated solid body.



Modeling View



Drawing member view

Why should I use it?

Use this feature when you want to preserve the face colors specified in modeling. Some companies assign face colors to indicate special manufacturing or design information. These colors must persist through the drawing process.

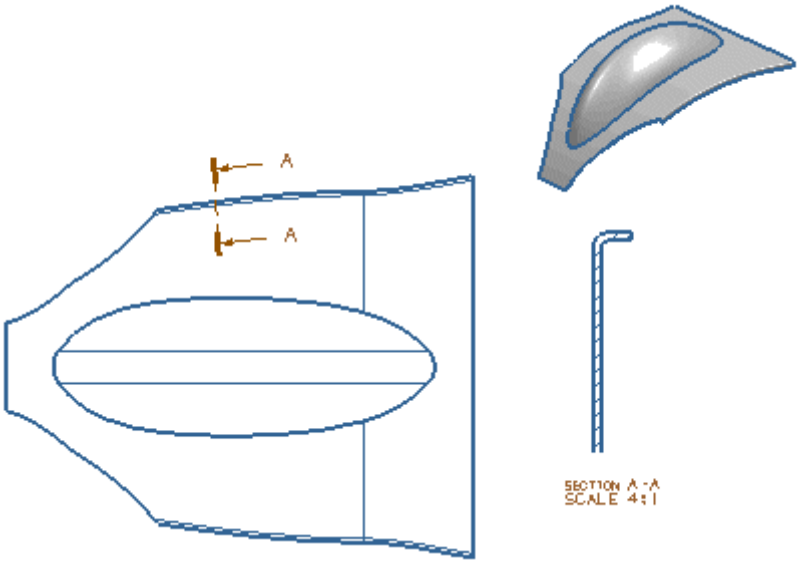
Where do I find it?

Application	Drafting
Toolbar	**Drafting Preferences® View Preferences**
Menu	**Preferences→View**
Shortcut Menu	Right-click a drawing member view border® **Style→General** page.

Location in dialog box	**General** page® **Wireframe Color Source** group® **From Face**

Oriented Section View

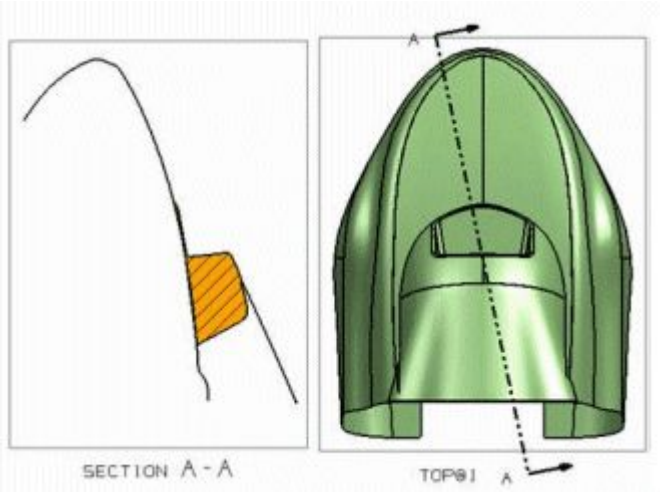
What is it?



3D oriented section cut

The simple 3D section line is displayed like a simple 2D section line but allows you to create a true 3D section cut based on the normal, binormal, and tangent vectors derived from a point on a curve.

- The section view is projected orthogonally from the parent view (that is, above, below, left, or right).
- The projection direction is determined by the arrow direction in relation to the parent view normal.
- The section view is oriented so that the cut lies on the view plane.



Why should I use it?

You can rapidly create a true section cut view without having to specify all the steps required to produce the same view using the Pictorial Section View tool.

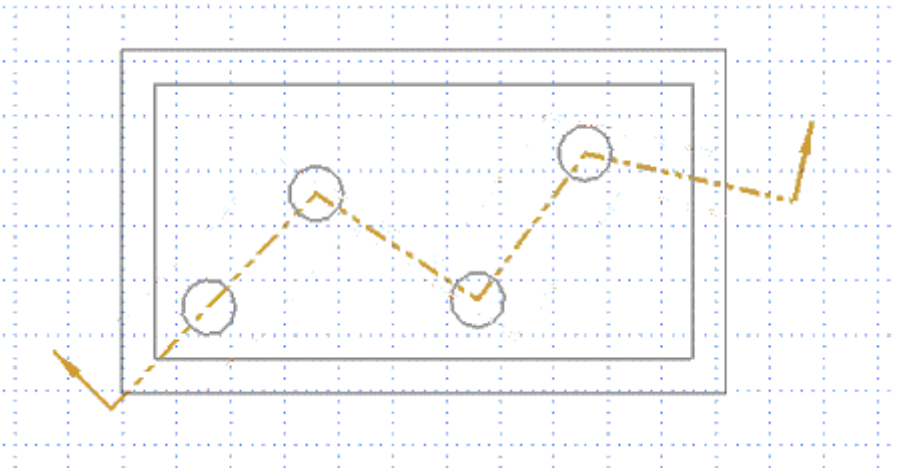
Where do I find it?

Application	Drafting
Toolbar	**Drawing® Section View**
Menu	**Insert® View® Oriented Section View**
Location in dialog box	**Section Line Creation® 3D Cut**

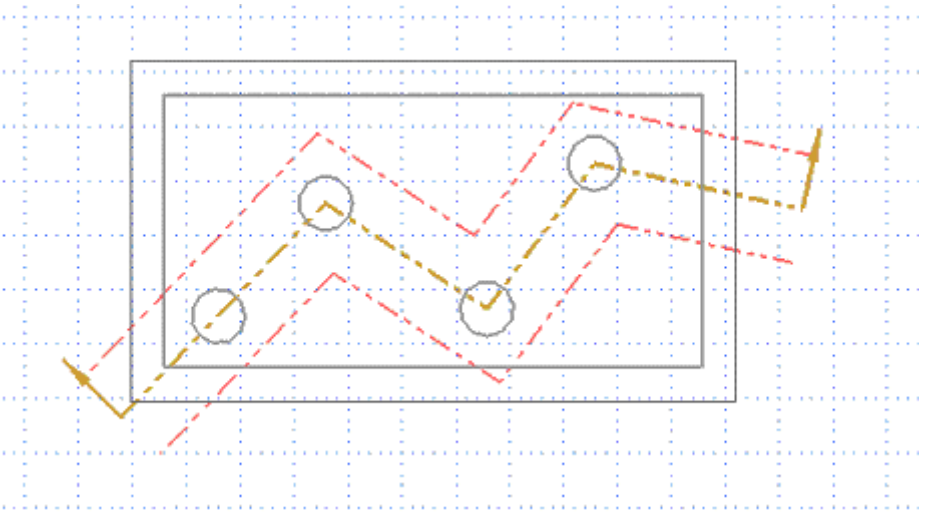
Offset section line

What is it?

You can create an offset section line on a section view that offsets normally to each section line segment and trims the offsets.



Before offset section line



After offset section line (Offset corridors only display during creation)

Why should I use it?

Use the feature when you only want to include a set of components within an offset corridor of a section line.

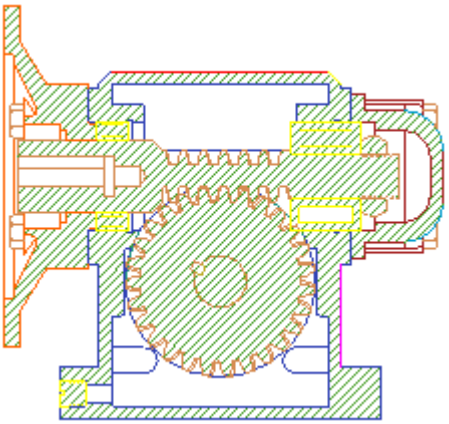
Where do I find it?

Application	Drafting
Toolbar	**Drawing**® **Section Line**
Menu	**Preferences**® **Section Line**
Location in dialog box	**Offset** group® **Use Offset**

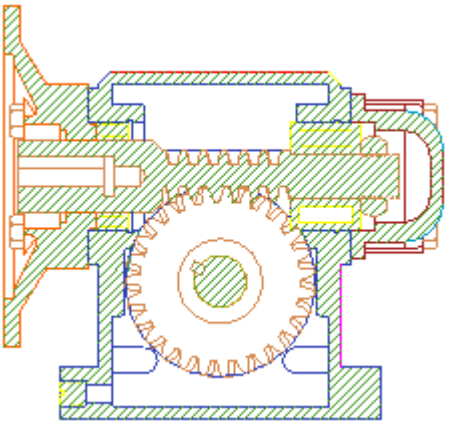
Sectioned/Non-sectioned solid bodies

What is it?

You can select solid bodies, along with components, as sectioned or non-sectioned in a drawing member view.



Sectioned



Non-sectioned using solid selection

Where do I find it?

Application	Drafting
Toolbar	**Drafting Edit® Section in View**
Menu	**Edit→View→Section in View**.
Location in dialog box	**Action** group® **Make Non Sectioned**

Datum terminator

What is it?

The new **Filled Datum Terminator** customer default controls what the setting is to be for the dialog the first time.

Filled datum terminator

Unfilled datum terminator

Where do I find it?

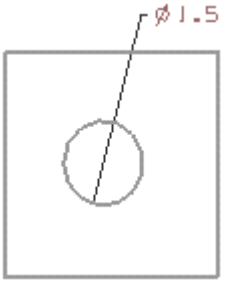
Menu	**File→Utilities→Customer Defaults**

Location in dialog boxes	**Drafting**→**General**→**Standard** page® **Customize Standard** button® **Annotation**® **Symbols** page® **Geometric Tolerance** group® **Filled Datum Terminator**

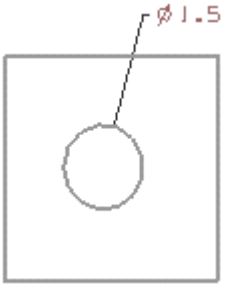
Infer diameter dimension for full circles

What is it?

Use this customer defaults option to specify that you want diameter dimensions created when you select holes or full arcs with the **Inferred** option on the **Dimension** toolbar.



Diameter dimension



Hole dimension

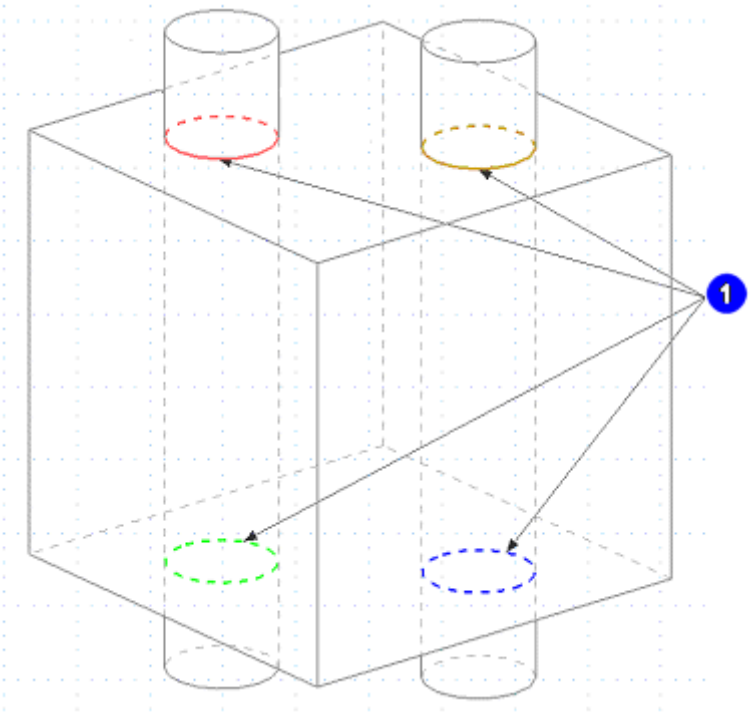
Where do I find it?

Menu	**File**→**Utilities**→**Customer Defaults**.
Location in dialog box	**Drafting**→**Annotation**→Dimensions page

Interference curves

What is it?

You can display an interference curve where an edge would be formed if two interfering solids were united. The interference curve is associative to the solids.



1 — **Interference curves**

Why should I use it?

You no longer have to manually draw interference curves for interfering solids.

Where do I find it?

Application	Drafting
Menu	Right-click a view border® **Style**.
Location in dialog box	**View Style→Hidden Lines** page® **Interfering Solids** group® **Yes, With Interference Curves**

Associativity for ordinate dimensions

What is it?

You can now establish an associativity for ordinate dimensions by selecting an extension line of a linear dimension. The ordinate dimension becomes associative to the geometry object to which the linear dimension is attached.

Why should I use it?

In circumstances where the density of geometry is high or the association is difficult to select, it is often easier to select a dimension that may have already established associativity to the geometry of interest.

Where do I find it?

Application	Drafting
Toolbar	**Dimension® Ordinate**
Menu	**Insert→Dimension→Ordinate**

Import Drafting Standard

What is it?

Import Drafting Standard lets you import Drafting Standards files you created in NX 5 into NX 6.

Why Should I Use It?

Use this feature when you want to preserve previous versions of your drafting customer defaults. A report is generated informing you of new, changed, and unchanged customer defaults.

Where do I find it?

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Drafting→General→Standard** page® select a standard® **Customize Standard** option® **Import Drafting Standard File** list box® select .dpv file

PMI

Regions

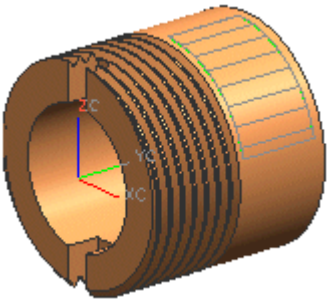
What is it?

You can now create PMI supplemental geometry regions that can be referenced by other PMI objects.

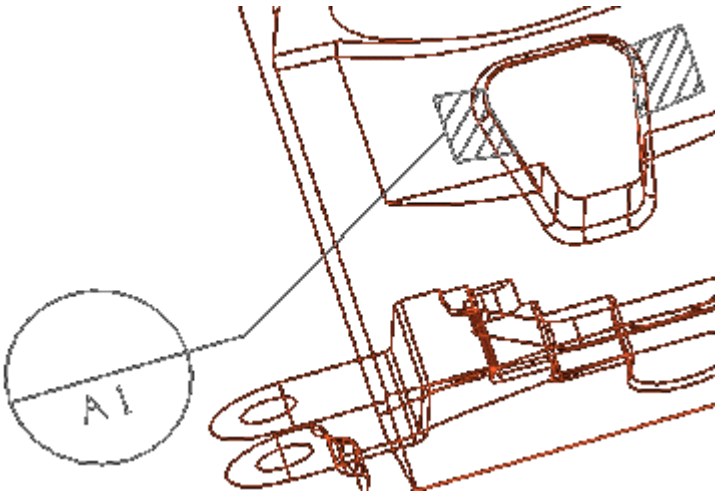
A supplemental geometry region:

- Can be circular, annular, rectangular, cylindrical, or arbitrary (using selected curves or faces). Initial values of rectangular, circular, and annular regions are controlled by customer defaults.
- Moves with a component when it is created associatively to that component.
- Can be referenced by multiple PMI objects.
- Appears in the **Part Navigator**.

The following figures are examples of two types of supplemental geometry regions.



Rectangular region that conforms to a face



Arbitrary region with 2 subregions (used as a datum target region)

You can:

- Specify whether a region should have crosshatching, and define the crosshatch attributes.

- Create supplemental geometry regions at the assembly level, as well as at the component level. If an assembly-level supplemental geometry region is based on geometry in an unloaded component, the supplemental geometry region is retained until its parent geometry is loaded.

You cannot:

- Drag or stack supplemental geometry regions.

- Place a supplemental geometry region in a reference set. PMI supplemental geometry regions are managed by PMI Assembly Filters instead.

- Cut, copy, or paste supplemental geometry regions.

If a PMI object terminates on a supplemental geometry region, the supplemental geometry region is automatically added to the PMI object's associated objects.

Why should I use it?

Supplemental geometry regions are typically used for designating target areas or limited region tolerance specifications.

Where do I find it?

Application	PMI
Toolbar	**PMI**® Datum and FCF ® **Region**
Menu	**PMI**® **Supplemental Geometry**® **Regions**

Bidirectional edits of PMI

What is it?

Modifications that you make to inherited PMI objects in a drawing are now automatically applied to the corresponding PMI objects in your model.

Note

Inherited checked GD&T (that is, the functions in **PMI**® **Geometric Tolerancing**) modifications are not applied.

Why should I use it?

This function lets you update your model while you are working in the context of a drawing.

Where do I find it?

Application	Drafting
Menu	**Edit**® **Annotation**® **Annotation Object**
Shortcut menu	Right-click an inherited PMI object in a drawing® **Edit**

Sheet Metal

NX Sheet Metal

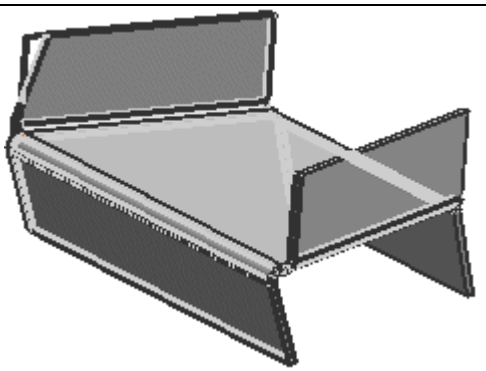
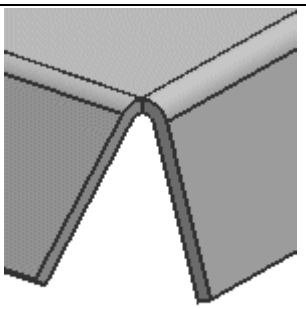
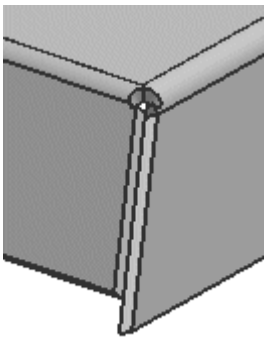
Closed Corner

What is it?

With **Closed Corner**, you can now create corners:

- Between flanges that bend in opposite directions.
- Between bends of different angles and different radii.
- With U-shaped, V-shaped, or rectangular cutout reliefs, in addition to the circular cutouts of previous releases.

You can also create reliefs without creating a corner.

Preview of a corner where the flanges bend in opposite directions.		
Before and after views of a corner with circular relief created on flanges at different angles		

Why should I use it?

This feature provides a convenient way to close the gap between two flanges. During flattening, the corner and relief geometry are made appropriate for a flat pattern.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**® **Closed Corner**
Menu	**Insert**® **Sheet Metal Feature**® **Closed Corner**
Location in dialog box	**Type** group® Type list® **Close and Relief** or **Relief** **Corner Properties** group® **Treatment** list

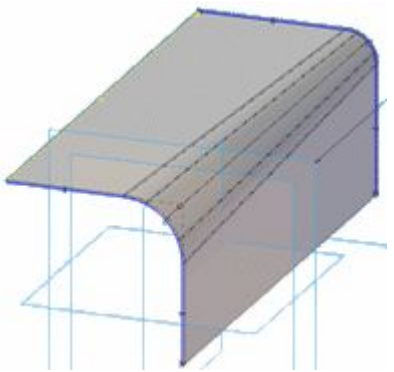
Multi-segment Lofted Flange

What is it?

Conical bend regions of a lofted flange can now be divided into up to 24 separate bend regions.

Each bend is represented as a bend centerline in the 3D model. In the flat pattern, these bend lines are displayed as centerlines in the visible edge color.

You can specify bend index marks when you save the sheet metal file as a flat pattern. These bend index marks are short lines connected to the edge of the part along the bend that help align the metal with the press brake. These marks only appear on the flat pattern.



Multi-segmented bend region

Why should I use it?

Some sheet metal manufacturing methods for creating conical bend regions use a series of cylindrical bends at angles to one another. This feature helps you design these bend regions without having to manually segment the section sketch curves. You simply specify the number of bend segments in the **Lofted Flange** dialog box.

Where do I find it?

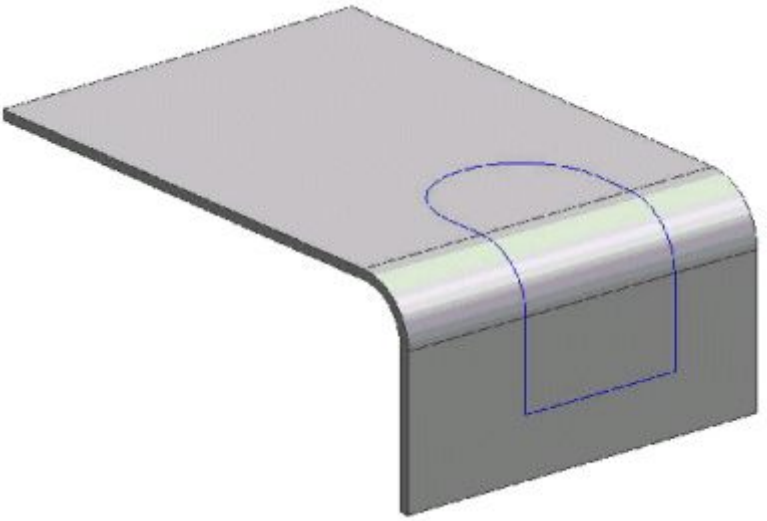
Application	NX Sheet Metal

Toolbar	**NX Sheet Metal® Lofted Flange**
Menu	**Insert® Sheet Metal Feature® Lofted Flange**
Location in dialog box	**Bend Segments** group® **Use Multi-segment Bends** check box

Normal Cutout

What is it?

You can now use 3D curves to create a Normal Cutout feature. Previously, you could only use planar curves.



Normal (3D) curves for use with Normal Cutout

Why should I use it?

When you create a Normal Cutout using a planar sketch, you cannot always control where the section curves are projected on the placement face. Using 3D curves gives you direct control over where the cuts are placed on the model.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal® Normal Cutout**
Menu	**Insert® Sheet Metal Feature® Normal Cutout**
Location in dialog box	**Type** group® Type list® **3D Curves**

Bend Taper

What is it?

You can now create tapers on bends and web regions of flanges and other similar features (such as contour flanges). You can:

- Taper the bend and web regions.
- Taper only the web region.
- Create tapers of different angles on each side of the web or bend region.
- Taper the bend and web regions at different angles.
- Select multiple bend regions to apply tapers to.
- Taper only one side of the bend or web and leave the other side unchanged.

Flange with bend and face tapers

Why should I use it?

Use this feature to easily remove material from bend and web regions.

Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal® Bend Taper**
Menu	**Insert® Sheet Metal Feature® Bend Taper**

Flat as solid without uniform thickness

What is it?

If you use thick plates in manufacturing and want to represent weld preparations on those plates, you can now:

- Create chamfers across bend regions.
- Apply **Unbend**, **Rebend**, **Flat Solid**, and **Flat Pattern** to those regions.
- Have those features be represented properly during forming and flat patterning activities.

Why should I use it?

The ability to create parts without uniform thickness allows you to have features such as chamfers along non-thickness edges, without losing geometry during an Unbend or a Rebend operation.

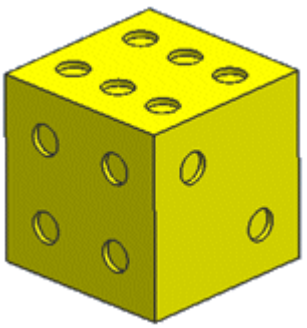
Where do I find it?

Application	NX Sheet Metal
Toolbar	**NX Sheet Metal**

Sheet Metal from Solid

What is it?

You can create a sheet metal part using a solid to model it. The solid represents the inner void of an enclosure. You select the faces from the solid to be panel (web) faces and edges between the faces to be bend regions. You specify bend properties, or use the ones you established in **NX Sheet Metal Preferences**. The software creates the model based on your inputs. You can then treat this part as you would any other sheet metal part.



Solid

Resulting sheet metal part

Why should I use it?

Complex geometry can make creating individual flanges cumbersome. **Sheet Metal from Solid** greatly simplifies and streamlines this process. It also provides a type of workflow familiar to users of I-deas Sheet Metal.

Where do I find it?

Application	NX Sheet Metal
Prerequisite	A solid part created using standard NX modeling techniques
Toolbar	**NX Sheet Metal® Sheet Metal from Solid**
Menu	**Insert® Sheet Metal Feature® Sheet Metal from Solid**

Flat Pattern

What is it?

You can now:

- Specify that the flat pattern have its exterior and interior corners rounded. When the cutting tool does not handle sharp corners in the tool path well, you can use this feature to create a flat pattern that does not have sharp corners. Many laser cutters, and similar tools such as plasma and water jet cutters, have to perform extra steps to make a sharp turn.

- Create the flat pattern without the curves that represent interior features. You can create a flat pattern uncluttered by the curves from beads, dimples, drawn cutouts, punch features, and louvers.

- Display bend center lines that go up differently than those that go down.

- Display all curves from non-uniform solids, where the top of the flat solid is not the same as the bottom. Curves on the bottom are displayed in the standard hidden line font and curves on the top are displayed in the same font as exterior curves. (Top and bottom are defined with respect to the face picked as the reference face when creating the flat pattern.)

Why should I use it?

Use these features to customize the flat pattern to suit your modeling and manufacturing requirements.

Where do I find it?

Application	NX Sheet Metal

Prerequisite	A sheet metal part as source for the flat pattern.
Toolbar	**NX Sheet Metal® Flat Pattern**
Menu	**Insert® Sheet Metal Feature® Flat Pattern**

Tool ID-Driven Bend Properties

What is it?

You can now use a Tool ID selected from a user-defined list to define the bend properties for your sheet metal part. This works in much the same way as the Materials Database selection introduced in NX 5. You add values to the BEND_TOOL_ID_TABLE.section of the *Sheet_Metal_Materials.txt* file, and NX populates the preferences dialog box with those values.

Why should I use it?

Different tooling creates different bend region behaviors. Some tools also have specific parameter values, such as radius. In these cases it may be convenient to create a table of tools and their associated radius, neutral factor, and material thickness values. You can then select an appropriate tool from the list to define the bending for the part.

Where do I find it?

Application	NX Sheet Metal
Menu	**Preferences® NX Sheet Metal**
Location in dialog box	**Part Properties® Parameter Entry® Tool ID Selection**

NX Sheet Metal part validation

What is it?

Two new validation routines have been added to the Check-Mate application specifically for validating areas of sheet metal parts.

- **Minimum Tool Clearance** validates that there is enough room between punch features and bend regions for tooling.
- **Minimum Web Length** validates the minimum desired length of flange webs.

You set the values for the checks in the **NX Sheet Metal Preferences** dialog box and run the checks in the Check-Mate application.

Why should I use it?

Just like any other part, sheet metal parts must be manufacturable. Creating checks with the proper values can help ensure that you are creating a final part that can be manufactured using the processes available to your company.

Where do I find it?

Minimum Tool Clearance and **Minimum Web Length**

Application	NX Sheet Metal
Menu	**Preferences® NX Sheet Metal**
Location in dialog box	**Sheet Metal Validation® Validation Parameters**

Check-Mate command

Application	NX Sheet Metal
Toolbar	**Check-Mate® Run Tests**
Menu	**Analysis® Check-Mate® Run Tests**
Location in dialog box	**Tests® Sheet Metal**

Chapter

8 Data Reuse

Reuse Library

Feature/Object template

What is it?

You can copy a frequently used feature or object from your model, and paste it as a Feature/Object template into a selected folder in the **Reuse Library**.

You can then reuse the feature or object by dragging the template from the **Reuse Library** into your current design.

Note

Teamcenter Classification folders are not supported.

Where do I find it?

Resource bar	**Reuse Library**
Location in palette	In the **Name** column, right click a folder, and choose **Add Reusable Object Here**.

Create a Feature/Object template

A Feature/Object template maintains a reference to the source part from which it is created. If the source part is in the same folder as the Feature/Object template, the reference information is saved in the source part. This is the recommended practice if you want your Feature/Object template to remain unchanged.

If the source part is not in the same folder as the Feature/Object template, the reference information is saved in an external KRX file. This means that if the source part is altered, the Feature/Object template may potentially change.

Do the following to ensure proper management of a Feature/Object template, and protect it from changes by unauthorized users:

1. Add the folder that contains your source part as the default Reuse folder:

Note

Skip this step if your company already has a default Reuse folder.

a. Choose **File→Utilities→Customer Defaults**.

b. In the **Customer Defaults** dialog box, select **Gateway→Reuse Library**, and click the **General** tab.

c. In the **Libraries Organized by Native Folder** box, enter the path to your Reuse folder.

d. Exit NX and restart the application

2. Open the file that contains the work part from which you want to copy objects.

3. In the graphics window, right-click the desired objects in the work part and choose **Copy**.

4. Click **File→New** and create a new file. Enter a name for the template in the **Name** box and enter the path to your Reuse folder in the **Folder** box. Click **OK**.

5. Choose **Edit→Paste** and paste the objects you selected in step 2 into your template file.

6. In the **Paste Feature** dialog box, **Select Face** is active. Select a face or plane and any other reference and click **OK**.

7. Save the file.

8. In the graphics window, right-click the pasted object and choose **Copy**.

9. On the Resource bar, click **Reuse Library** and expand the **Reuse Library** node.

10. Right-click your Reuse folder and choose **Add Reusable Object Here**.

    The selected object is now added to your Reuse folder and will not change if changes are made to your source object.

Filter Member Select view

What is it?

You can select viewing options and filter information in the **Member Select** panel of the **Reuse Library**.

You can:

- Select a viewing option, for example, thumbnail view or icon view, from the View Type list.
- Filter the view by selecting the items you want, for example, KE parts or Feature/Object Templates, from the Filter View list.

Where do I find it?

Resource Bar	**Reuse Library**
Location in palette	**Member Select** panel® View Type list and Filter View list .

Reuse Library-Fastener Assembly

What is it?

You can now add fasteners (bolts, screws, nuts, and washers) into selected holes automatically.

The fasteners are retrieved from the **NX Machinery Library**, which has a large variety of ANSI Inch, Metric, ISO, JIS, and other standard hardware. The fasteners are fully parameterized NX parts.

When you create a hole using the **Hole** command, the screw/bolt in the Fastener Assembly is automatically selected according to the hole diameter and depth.

When you create a hole using the pre-NX 5 **Hole** command, the default screw/bolt in the Fastener Assembly template is assumed for all holes.

You can use:

- **Edit Reusable Component** to change the type, size, length, flip, and other editable dimensions of the standard components in the Fastener Assembly.
- **Fastener Assembly Configuration** saves the configuration or structure of the fastener assembly as a Fastener Assembly template.

Why should I use it?

In the **Fastener Assembly Configuration** dialog box you can pre-select a fastener assembly structure that is a bolt, washer (top stack), and nut (bottom stack) to be used on different holes. You can save the configurations for future reuse or you can distribute them for sharing. When loading a fastener assembly, you can select the saved configuration, as well as load a fastener assembly to be used on many different holes at the same time. The corresponding fastener assembly configuration can be reused on each hole type automatically.

Where do I find it?

Menu	**Tools→Reuse Library→Fastener Assembly**
Toolbar	**Reuse Library**

Shape Search-Geolus integration

What is it?

The new **Shape Search** command uses Geolus shape recognition technology. Use this command to locate parts of a particular shape and size in the Geolus database that is setup by your system administrator.

Note

This functionality will only be available with an "NX Integration to Geolus" license.

You can search for parts by:

- Geolus attributes such as part numbers or materials.
- Solid or sheet bodies.
- Part shape similarity or part size range.

Why should I use it?

Shape searching saves you valuable design time by helping you quickly find existing parts for reuse.

Where do I find it?

Application	Available in all applications except Gateway.
Menu	**Tools→Shape Search**
Location in dialog box	**Define Search→Type**

Chapter

9 Systems Design

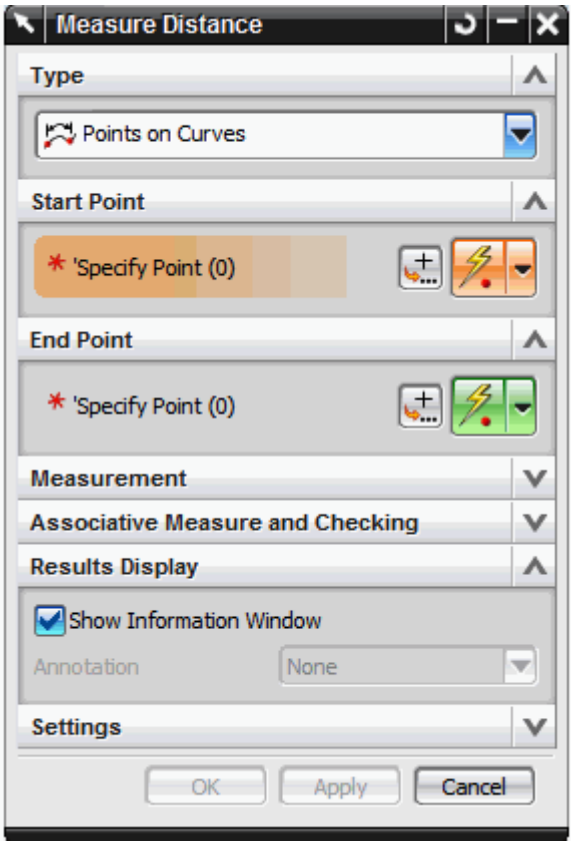
Routing Systems

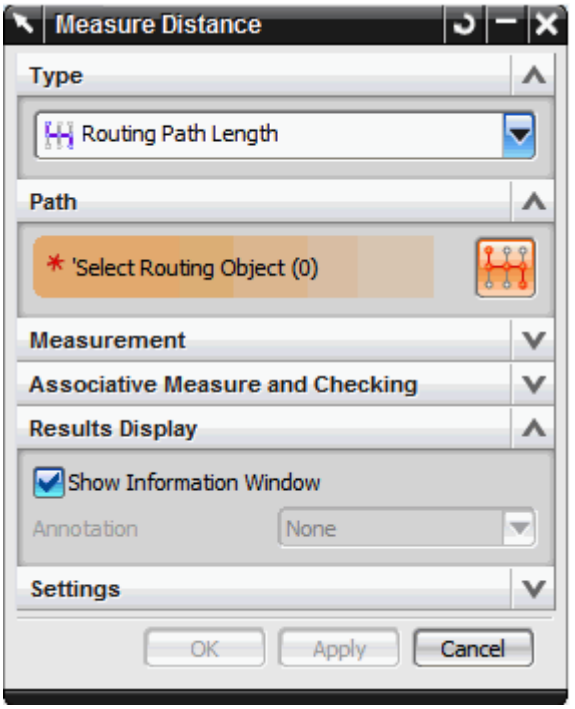
Routing Systems: General

Measure Distance

What is it?

Distance measurement with **Type of Points on Curves** or **Routing Path Length** gives you distance between points on a set of curves or the length of a routing path.





Why should I use it?

Use **Measure Distance** to get the information about the length of a path.

Where do I find it?

Application	All NX applications
Menu	**Analysis→Measure Distance**
Toolbar	**Utility**

Minimum bend radius spline violations

What is it?

A new design rule capability highlights specific violation locations for splines violating the minimum curve bend radius design rule.

Why should I use it?

Use this new capability to find the exact location of sharp corners on splines to help edit or eliminate the problem.

Where do I find it?

For interactive tests:

Application	Routing Electrical / Routing Mechanical
Menu	**Analysis→Check Mate→Run Tests** , and **Analysis→Design Rules→Interactive Check**

For automatic checks:

Application	Routing Electrical
Menu	**Preferences→Wiring**
Location in dialog box	**General** page® **Report Routing Errors** check box.

Smart Router: Quick Path

What is it?

Quick Path automatically creates a collision free path between two specified locations, and optionally, through specified intermediate locations. The resulting path comprises lines and arcs, but not splines, and avoids obstacles by a specified clearance value. You can adjust the path for the final desired path.

Why should I use it?

Use **Quick Path** to avoid manually creating a collision-free non-trivial path between two locations.

Where do I find it?

Application	Routing Electrical / Routing Mechanical
Menu	**Insert→Routing Path→Quick Path**
Toolbar	**Routing Path→Quick Path**

Smart Router: Transform Path

What is it?

Transform Path is a tool that transforms or copies Routing objects. The new version of **Transform Path**:

- Is more dynamic and allows a wider variety of transformations.
- Supports collision detection.

Why should I use it?

Use **Transform Path** to quickly and easily transform Routing objects. Select Routing objects in the graphics window and transform them to the desired location. Any interferences display dynamically as they occur.

Where do I find it?

Application	Routing Electrical / Routing Mechanical / Routing Logical
Menu	**Edit→Routing Path→Transform Path**
Toolbar	**Routing Path→Transform Path**
Shortcut menu	In the graphics window, right-click a routing object, and choose **Transform Path**.

Smart Router: Spline Path

What is it?

Spline Path is a convenient tool for creating and editing routing splines. These splines may be constrained to components or other Routing geometry to create highly associative systems.

Why should I use it?

Use **Spline Path** to create routing splines that model flexible stock such as hoses, cables or harnesses. These splines may also have their length locked to ensure design stability. In addition, splines may be shaped to mimic gravity (slack and locked length with slack).

Where do I find it?

Application	Routing Electrical / Routing Mechanical / Routing Logical
Menu	**Insert→Routing Path→Spline Path**
Toolbar	**Routing Path→Spline Path**
Shortcut menu	In the graphics window, right-click a routing spline, then choose**Edit Spline**.

Space Reservation

What is it?

Space Reservation is a command which allows you to quickly and easily assign a new space reservation stock on the selected segments.

Why should I use it?

This tool provides an easy and quick way to add space reservation stock. You can use **Edit Stock** to edit space reservation stock.

Where do I find it?

Application	Routing Electrical / Routing Mechanical
Menu	**Insert→Routing Stock→Space Reservation**
Toolbar	**Routing Stock→Space Reservation**
Shortcut menu	Right-click one or more segments, then choose **Space Reservation** or **Edit Space Reservation**.

Part Family tools

What is it?

Routing ports can be attached to port features in **Qualify Part**, and port names are now controlled by expressions. This allows you to control routing ports with Part Family spreadsheets.

Why should I use it?

Use this when you need variable numbers and names of routing ports. For example, when you design a connector with 20 terminal ports and would like to create similar connectors with 1 terminal port, 10 terminal ports and/or 5 terminal ports, with different port names.

Where do I find it?

Application	Routing Electrical / Routing Mechanical
Menu	**Tools→Part Families**

Global changes to dialog boxes

What is it?

Routing has updated numerous dialog boxes to increase productivity by:

- Increasing the discoverability of commands by providing a more logical layout in the dialog boxes.
- Reusing common dialog box elements across similar commands to achieve consistency of the user interface.
- Remembering the dialog box settings to gradually adapt NX to your way of working.
- Managing the location of dialog boxes and toolbars so that the graphics window has minimal obstruction.

Routing Systems: Electrical

Connector redundancy design rule

What is it?

This is a design rule which can be setup to run automatically when you import electrical data into the **Component List Navigator**. It checks if two or more connector rows have the same values for the columns defined in the user preference in the Application View file.

You can use the **Interactive Check** command to run the check manually.

Why should I use it?

Use this design rule to check for redundant connector rows in the **Component List Navigator** after importing electrical data into NX.

Where do I find it?

For manual checking:

Application Menu	Routing Electrical
	Analysis→Check Mate→Run Tests , and
	Analysis→Design Rules→Interactive Check

Copy Overstock

What is it?

Copy Overstock allows multiple designers to work on the same harness where that harness is spread across multiple subassemblies.

Why should I use it?

Use **Copy Overstock** when overstock defined at a subcomponent level to the harness assembly needs to be recreated at the harness assembly.

Where do I find it?

Application	Routing Electrical
Menu	**Edit→Routing Electrical→Copy Overstock**
Toolbar	**Routing Stock→Copy Overstock**

Abort Automatic Routing

What is it?

Abort Automatic Routing utilizes the **Work in Progress** dialog box to allow you to interrupt a wire routing operation.

The **Work in Progress** dialog box automatically appears during an **Automatic Routing** operation.

Auto Place Connectors

What is it?

The **Auto Place Connectors** option automatically finds and places connector(s) based on information in the **Component List** entry for the corresponding logical connector.

Why should I use it?

Auto Place Connectors is an efficient way to automatically add connector parts to the harness assembly and place them on the correct port of the device/equipment.

Where do I find it?

Application	Routing Electrical
Shortcut menu	Select one or more nodes in the **Component List Navigator**, then right-click **Place Connectors**.

Options and Variants Management

What is it?

Options and Variants Management in Routing Electrical is a method to describe variant electrical content with an NX routing harness assembly.

Why should I use it?

Use **Options and Variants Management** to define options, conditions, rules, and configurations for a harness assembly. Connections and logical connectors can be configured in and out of the design according to design or product requirements.

Where do I find it?

Application	Routing Electrical
Toolbar	**Routing Electrical→Options and Variants Management**
Short cut menu	Right-click **Options and Variants Management** from the **Connection List** and/or **Component List**.

Clarify Stock Picks

What is it?

Clarify Stock Picks gives you more detailed information about the type of stock that you select. This is in addition to the information provided by **Routing Stock Selection**.

Netlist Content History

What is it?

Netlist Content History logs differences between imported data and existing data and stores them with the work part for later reference.

Why should I use it?

Netlist Content History can provide details on when specific changes to the connection and component lists took place.

Where do I find it?

Application	Routing Electrical

Toolbar	**Wiring Tools→Netlist Content History** . To erase the netlist history, select **Erase Netlist History** .

Wire Routing Stock Style

What is it?

Wire Routing Stock Style causes wires to be routed with centerline stock while not affecting the creation of other stocks in detailed or simple mode.

Why should I use it?

Auto or manual routing with centerline stock increases performance of routing wires.

Where do I find it?

Application	Routing Electrical
Menu	**Preferences→Wiring→Navigators→ Stock Style**, and **Customer Defaults→Navigators→ Stock Style**
Shortcut menu	Find the action to change the stock style of a harness under the right-click menu on a harness node in the **Connection List**.

PLM XML Viewer

What is it?

The **PLM XML Viewer** is a stand alone application outside of NX that lets you view the contents of a Routing Electrical PLM XML file in the format of a connection and component list and a topology breakdown.

Why should I use it?

The **PLM XML Viewer** allows quick visibility to the data present in an electrical PLM XML file.

Where do I find it?

Application	Routing Electrical
Directory	Go to the `utils` directory under the *UGROUTE_ELEC* kit, select **PLMXML Viewer.exe**.

Automotive Applications

General Packaging

Pedestrian Protection

What is it?

Use the **Pedestrian Protection** command to create the head and leg impact areas. You can offset, project, or calculate the intersection areas to evaluate whether the current design meets the pedestrian protection regulations of the NCAP (New Car Assessment Program).

The following assessment programs are supported:

- Global Technical Regulation
- European
- Japanese
- Korean

Note

When creating the head impact area, if you use the Global Technical Regulation assessment program, you can evaluate whether the current design meets the pedestrian protection requirements in the early design stage. If you use the other assessment programs, you can provide the head impact area to the automobile manufactures for NCAP testing.

A new **Pedestrian Protection** customer default option enables you to customize the following:

- Values and settings defined within each NCAP
- Color, layer, and font for each output item
- Initial values for other inputs of the user interface

Why should I use it?

Pedestrian Protection helps you save design time as the manual process for evaluating the current designs against NCAP standards is time consuming, tedious, and error-prone.

Where do I find it?

Pedestrian Protection command

Application	Modeling
Toolbar	**General Packaging® Pedestrian Protection**
Menu	**Tools® Vehicle Design Automation® General Packaging® Pedestrian Protection**

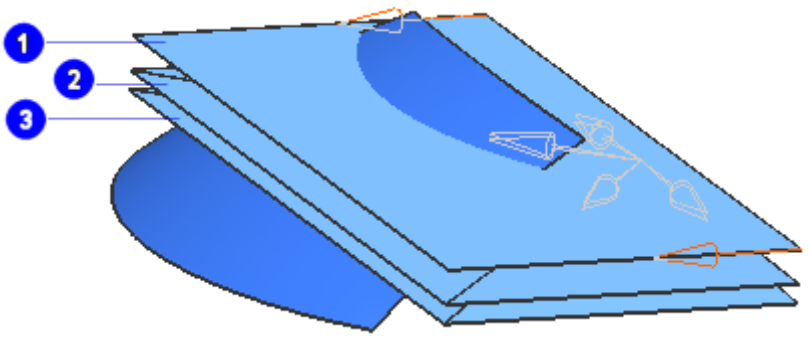
Pedestrian Protection customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Customer Defaults® Vehicle Design – General Packaging® Pedestrian Protection**

Vision Planes

What is it?

Use the **Vision Planes** command to create vision planes starting from the V-points, between which only specially defined vision obstructing elements can be positioned. The upper plane starts at the top V-point and is parallel to the road surface. The lower planes start from the bottom V-point and are inclined towards the front and the sides.



Vision Planes feature

1. Upper plane
2. Middle plane
3. Lower plane

Why should I use it?

You can use this command to:

- Create vision planes and perform vision study requirement defined for 77/649/EEC and Chinese GB11562-1994 standards.

- Find the location of vision planes, which can be used in an early phase of development to help in the interior design process, especially instrument panel design.
- Validate whether only allowed elements are located in the area defined by the vision planes.

Where do I find it?

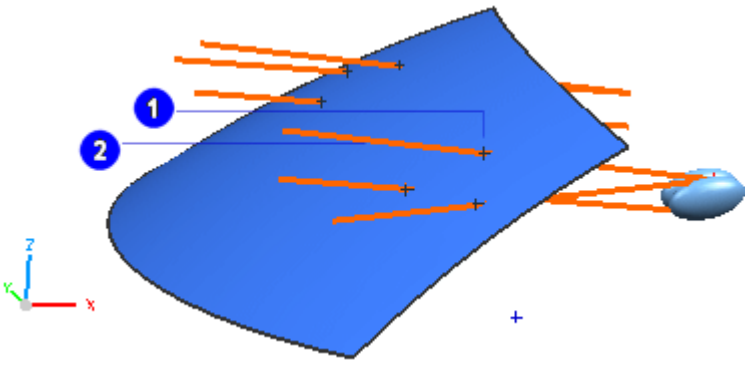
Application	Modeling
Prerequisite	You must know the positions of V points, or, it must be possible either to create EEC vision points based on SgRP, or create an eyellispse using the **Eyellipse** wizard.
Toolbar	**General Packaging® Vision Planes**
Menu	**Tools® Vehicle Design Automation® General Packaging® Vision Planes**

Windshield Datum Points

What is it?

Use the **Windshield Datum Points** command to create six datum points and the corresponding rays on the transparent area of the windshield as defined by 77/649/EEC and the Chinese GB11562-1994 standards.

You must provide the windshield geometry to get the exact locations of these six datum points. Otherwise, you can only view the corresponding rays.



1 Windshield datum point

2 Corresponding ray

Why should I use it?

The position of these windshield datum points can be used as a requirement during windshield design, or to validate an existing windshield design.

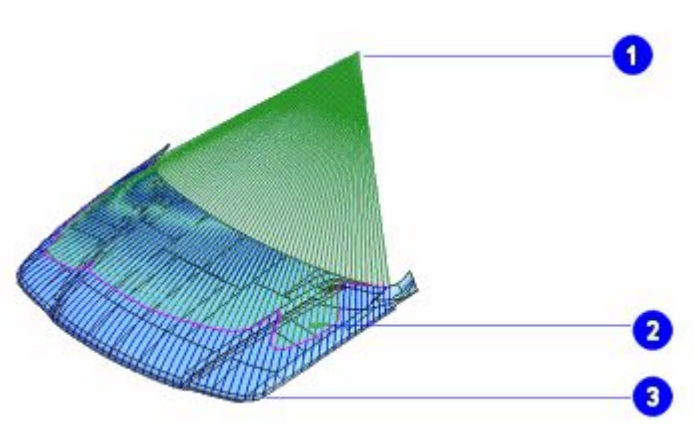
Where do I find it?

Application	Modeling
Prerequisite	You must know the positions of V points, or, it must be possible either to create EEC vision points based on SgRP, or create an eyellipse using the **Eyellipse** wizard.
Toolbar	**General Packaging® Windshield Datum Points**
Menu	**Tools® Vehicle Design Automation® General Packaging® Windshield Datum Points**

Hood Visibility Line

What is it?

Use the **Hood Visibility Line** command to calculate a line on the hood which represents the limit of the driver's visible hood area.



Hood Visibility Line

1. Eye point
2. Hood Visibility Line
3. Hood surface

Why should I use it?

You can use this command to:

- Know the visible and invisible area on a hood from the driver's eye point.
- Evaluate the direct view area of the hood at a product planning level required by certain regulations, for example EEC90/630 EU.

Where do I find it?

Application	Modeling
Prerequisite	You must know the positions of V points, or, it must be possible either to create EEC vision points based on SgRP, or create an eyellipse using the **Eyellipse** wizard.
Toolbar	**General Packaging® Hood Visibility Line**
Menu	**Tools® Vehicle Design Automation® General Packaging® Hood Visibility Line**

Chinese and EEC standards

What is it?

Options added to support Chinese standards include:

- **China GB15084 – 1994** in the **Mirror Certification** wizard.
- **GB - Chinese Standard** in the **Windshield Vision Zones** and **A-Pillar Obstruction** wizards.
- **User Defined** in the **2D Manikin** wizard. This supports customized human size for Chinese and Japanese manikins.

These options are added to the **Standard** step in the wizards.

Where do I find it?

Application	Modeling
Toolbar	**General Packaging**
Menu	**Tools® Vehicle Design Automation® General Packaging**

Allow Mirror Rotation

What is it?

This new option in the **Mirror Certification** wizard lets you specify whether to let NX adjust the mirror to an optimal angle.

Where do I find it?

Application	Modeling
Toolbar	**General Packaging® Mirror Certification**

Menu	**Tools® Vehicle Design Automation® General Packaging® Mirror Certification**
Wizard	**Mirror Profile** selection step

Requirements Checks

What is it?

Checks in General Packaging are now automatically converted to Requirements Checks for the following wizards:

- **Direct Field of View**
- **2D Manikin** (**Driver Posture** and **Manikin Joint Angle**)
- **Vehicle Packaging – Driver Posture**
- **A Pillar Obstruction Angle**
- **Mirror Certification**
- **Windshield Vision Zones**

You must define the internal or external sources for the requirements and specify the source to use. See Customer Defaults for Requirements.

Where do I find it?

Application	Modeling
Toolbar	**General Packaging**
Menu	**Tools® Vehicle Design Automation® General Packaging**

View Requirements Checks results

What is it?

You can now view Requirements Checks status and results for a number of wizards in:

- The **Check** step in the **Direct Field of View** wizard, which displays the Checker status.
- The **Validation Results** dialog box when you click **View Validation Results**.
- The Check-Mate results log file in an Internet Explorer window when you click **Generate Validation Log file** .

- The **Check Requirements** dialog box when you double-click **Direct_Field_of_View** in the **Part Navigator**.

These viewing methods apply to the following wizards:

- **Direct Field of View**
- **2D Manikin** (**Driver Posture** and **Manikin Joint Angle**)
- **Vehicle Packaging — Driver Posture**
- **A Pillar Obstruction Angle**
- **Mirror Certification**
- **Windshield Vision Zones**

Why should I use it?

You can access the validation results in the RDDV (Requirement Driven Design Validation) framework in an easy and consistent way.

Customer defaults for Requirements

What is it?

The following **Vehicle Design - General Packaging** customer defaults have been added to help you define the default Requirements values. These Requirements values apply to the following wizards:

- **Direct Field of View**
- **2D Manikin** (**Driver Posture** and **Manikin Joint Angle**)
- **Vehicle Packaging - Driver Posture**
- **A Pillar Obstruction Angle**
- **Mirror Certification**
- **Windshield Vision Zones**

Customer default option	Description
Requirements - Setting	Lets you specify the requirement source type to be used for checks.
Requirements - Pre-set Values	Lets you define the pre-set values for every wizard. These values are used when you use Pre-Defined Standard as the requirement source for checks.

Requirements in Teamcenter	Lets you specify the requirements item ID/Revision for General Packaging checks in Teamcenter 2007.
Requirements in Other Sources	Lets you specify the project name and property names for the requirements. These names are used by the following source types: spreadsheet, XML, and Teamcenter Systems Engineering.

Why should I use it?

You can create your requirements outside NX, and specify your source documents in NX, in the **Customer Defaults** dialog box.

Where do I find it?

Application	Modeling
Toolbar	**General Packaging**
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Vehicle Design - General Packaging**

Die Design

Die Shoe

What is it?

Die Shoe is a new command in **Die Design** that creates a die shoe which supports the interior die design castings.

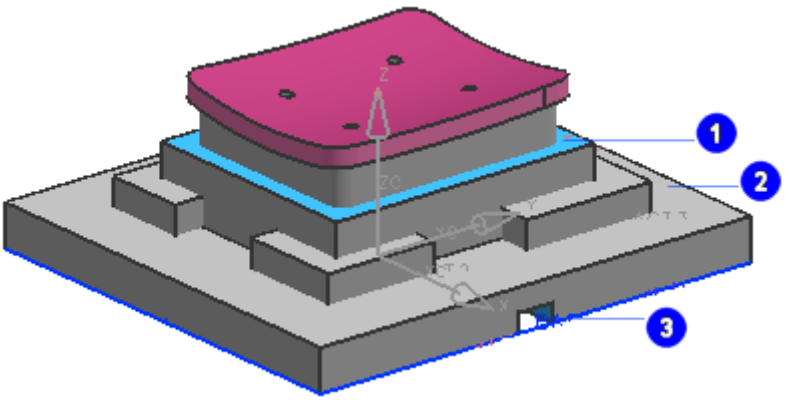
You can:

- Design the base casting of a die shoe which provides the framework for mounting and aligning castings in the complete die set.

- Specify pertinent data such as the base plane, flange profile, and the deck outline internal to the die shoe feature.

  Note

  To create the die shoe, you must have an idea of the size of your press operation box and the envelope of the internal data.



Die Shoe feature

1. Main deck
2. Base flange
3. Centerline slot

Why should I use it?

Use the **Die Shoe** command to create the basic die shoe which supports the interior die design castings. The core shoe casting can be later enhanced using either manual modeling methods or other Die Design commands.

Where do I find it?

Application	Modeling
Toolbar	**Die Design® Die Shoe**
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Die Shoe**

Sub features

You can use the following new commands to create internal Die Design sub-features that appear in the history of the model in the **Part Navigator**:

Cast Relief

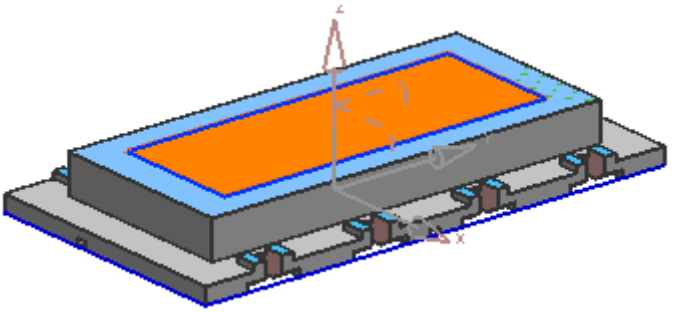
What is it?

Use the new **Cast Relief** command to specify all reliefs to be added to the casting.

When you use the **Cast Relief** command:

- You must create a closed profile for each relief, as each relief is specified by a closed profile. You can create multiple closed profiles in the casting.

- If you select one or more closed profiles for a Cast Relief feature, each closed profile defines a solid body.
- If you specify a target body, relief bodies are united to the target body.



Cast Relief feature

Where do I find it?

Application	Modeling
Toolbar	**Die Design® Cast Relief**
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Sub-features® Cast Relief**

Clamping Slot

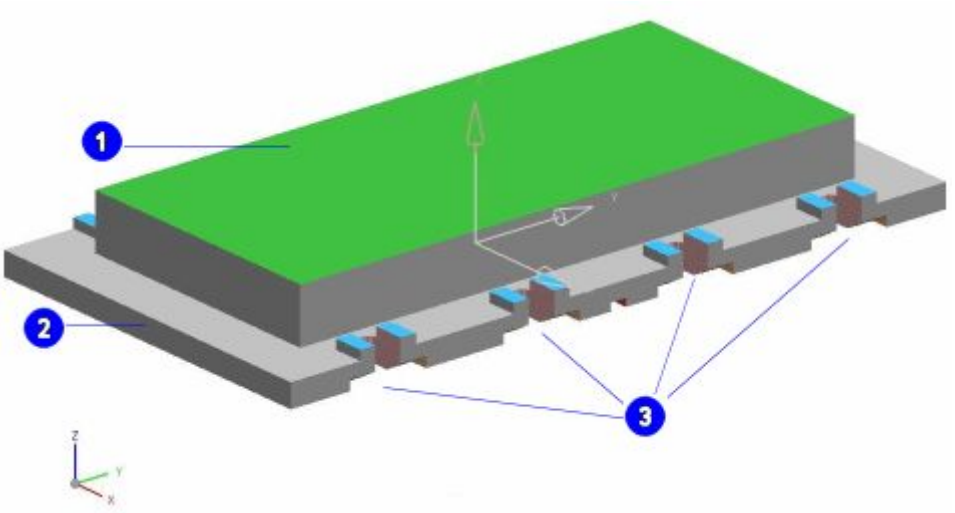
What is it?

Use the new **Clamping Slot** command to create a clamping slot on a casting.

When you use the **Clamping Slot** command:

- You can create **Hydraulic**, **Traveling**, or **Automatic** types of slots, depending on the clamping situation.
- You must specify a location for the Clamping Slot feature. You can specify multiple locations along a flange.
- If you specify the target body, a Clamping Slot feature is created at each specified location and is united to the target body.
- If you do not specify target bodies, clamping slots are created as individual solid bodies.

The following graphic shows a simple die shoe to which a Clamping Slot feature is added.



1 Main deck

2 Base flange

3 Clamping Slot feature

Where do I find it?

Application	Modeling
Toolbar	**Die Design® Clamping Slot**
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Sub-features® Clamping Slot**

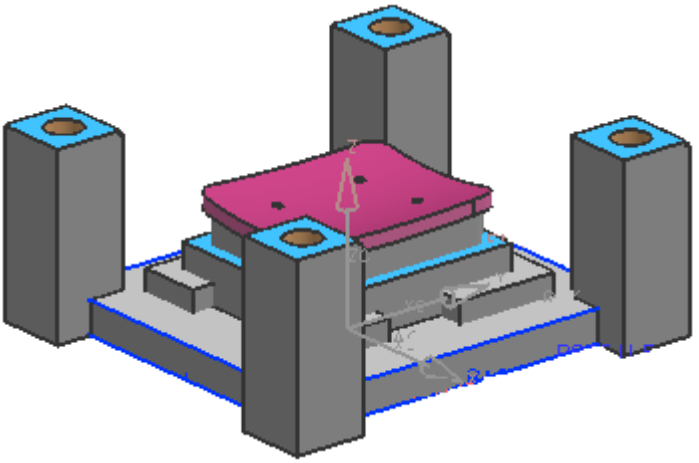
Heelpost

What is it?

Use the new **Heelpost** command to specify a single heelpost to be added to the casting.

When you use the **Heelpost** command:

- You must define the guide pin hole if you choose the **Guidepost/Wearplate** type. You need not create guide pin holes for the **Storage** or **Safety Block** types.
- You must specify the placement of the center of the post.
- If you specify the target body, a Heelpost feature is created at each specified location, and is united to the target body.
- If you do not specify target bodies, heelposts are created as individual solid bodies.



Heelpost features

Where do I find it?

Application	Modeling
Toolbar	**Die Design® Heelpost**
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Sub-features® Heelpost**

Keyway

What is it?

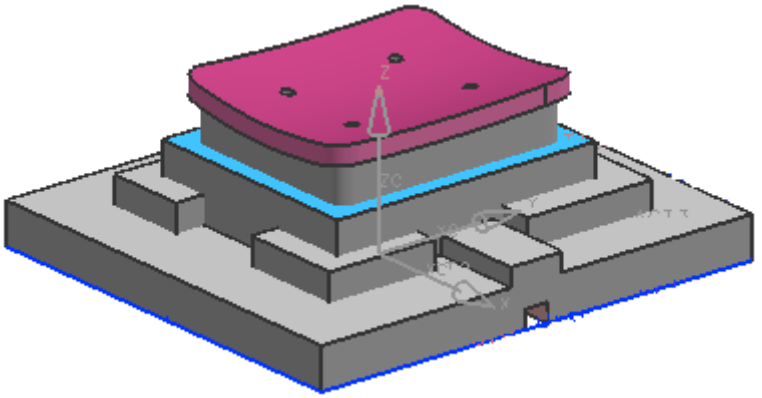
Use the new **Keyway** command to create a single Keyway feature on a casting. The type of keyway specified determines how the machining runoff must be constructed.

When you use the **Keyway** command:

- The type of keyway you select determines how the machining runoff is constructed. Rectangular keyways can be used for cutting across a rib for the runoff, whereas circular keyways can be used for plunge cutting into a casting area.

- If you specify the target body, the Keyway feature is united to the target body.

- If you do not specify target bodies, the Keyway feature is created as an individual solid body.



Keyway feature

Where do I find it?

Application	Modeling
Toolbar	**Die Design® Keyway**
Menu	**Tools® Vehicle Manufacturing Automation® Die Design® Sub-features® Keyway**

Die Engineering

Compensate Rough Data

What is it?

Use the **Compensate Rough Data** command to create a coarse, rough, or fine copy of curves, edges, and sheet bodies in your input data.

You can then start the Die Engineering and Die Design processes using this simplified form of the input data early in the product life cycle even if the initial product data has overlapping faces and gaps in the sheet body.

When the initial data is finalized, you can request an exact copy, and the subsequent operations utilize the copy of the final data in their computations.

In the following graphic, the dotted curve is the original curve and the solid curve is the compensated curve.

Coarse Rough Fine

Where do I find it?

Application	Modeling
Toolbar	**Die Engineering/ Die Design/ General** **Packaging® Compensate Rough Data**
Menu	**Tools® Vehicle Manufacturing Automation® Die Engineering/ Die Design® Compensate Rough Data** **Tools® Vehicle Design Automation® General Packaging® Compensate Rough Data**

Die Validation

Die Validation is enhanced to increase functionality and improve usability. New commands are added and existing commands are enhanced to better support press line simulation.

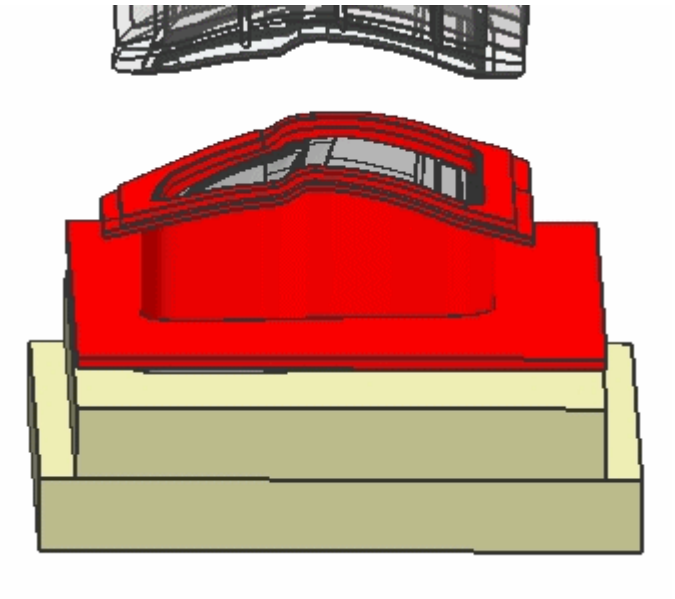
New commands

Cushion Programming

What is it?

Cushion Programming is a new command that enables you to modify the cushion parameters of any operation that has a programmable cushion. The attached Press Model determines which operations have a programmable cushion.

The following animation shows a preview of the motion cycle of the cushion.



1. Locked
2. Begin cushion lift
3. End cushion lift

Where do I find it?

Application	Modeling
Toolbar	**Die Validation® Cushion Programming**
Menu	**Tools® Vehicle Manufacturing Automation® Die Validation® Cushion Programming**

Transport Curves

What is it?

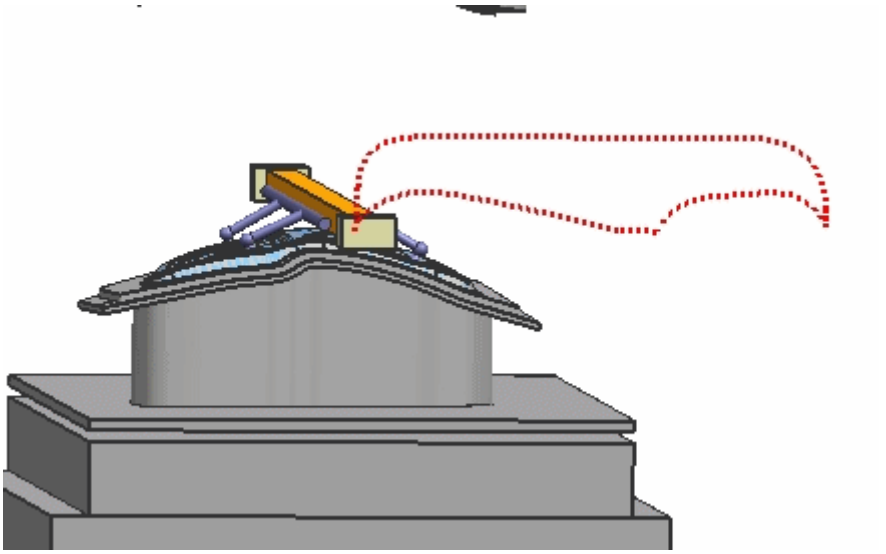
Use the **Transport Curves** command to modify the transport curves of any operation that has a transport device. The attached Press Model determines which operations have a transport device.

Each transport device:

- Contains a default set of transport curves. Some press models can contain extra sets of curves that can replace the default curves. You can also import your own set of transfer curves from a comma separated values (CSV) file.

- Can have one or more transport devices and each device can have one or more axis of movement. Every axis has a transport curve that defines how

it moves during the simulation. You must replace the entire set of curves (one for each axis) for an operation, to modify the transport motion.



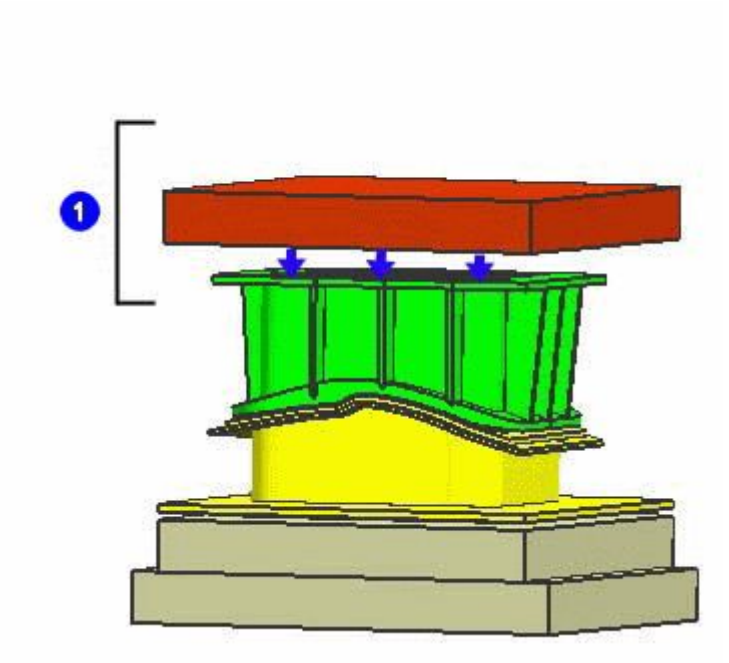
Where do I find it?

Application	Modeling
Toolbar	**Die Validation® Transport Curves**
Menu	**Tools® Vehicle Manufacturing Automation® Die Validation® Transport Curves**

Slide Height Adjustment

What is it?

Use the **Slide Height Adjustment** command to modify the slide height of any operation that has a slide. The attached press model determines which operations have an adjustable slide.



1 Slide height range

Why should I use it?

Use this command to adjust the slide to match the height of the dies that are mounted to it. Adjustment is restricted to the axis of the slide motion and limited to a range determined by the press model.

Where do I find it?

Application	Modeling
Toolbar	**Die Validation® Slide Height Adjustment**
Menu	**Tools® Vehicle Manufacturing Automation® Die Validation® Slide Height Adjustment**

User Defined Motion

What is it?

User Defined Motion is a new command that enables you to add motion to the press line simulation. You can apply motion to any user data mounted to the press model. This motion is in addition to the motion of the press model.

You can define:

- Linear movement along a defined vector using the **Linear** option.
- Angular movement around a defined axis using the **Rotary** option.

The motion curve is a set of 360 values (one for each timing angle 0-359) that determines how far to move along or around the axis at each point of the simulation.

You can accept the simple motion curve that is generated or you can supply your own curve values from a comma separated values (CSV) file.

Where do I find it?

Application	Modeling
Toolbar	**Die Validation® User Defined Motion**
Menu	**Tools® Vehicle Manufacturing Automation® Die Validation® User Defined Motion**

Enhanced commands

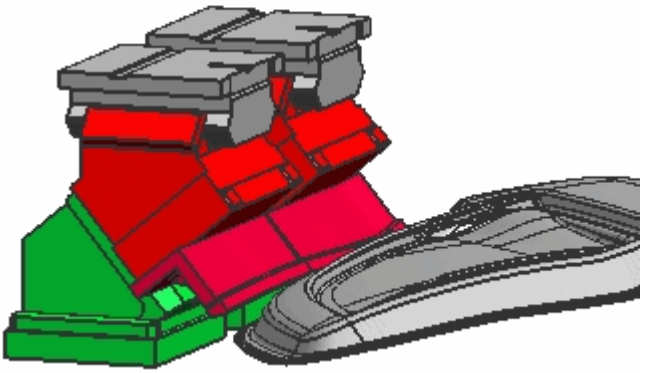
Linear Cam

What is it?

You can now specify the contact faces to calculate the movement of the cam.

You can:

- Preview the cam cycle that reflects the current settings stored in the cam in the graphics window.
- Recalculate and update a selected cam's motion using the current bodies.
- Recalculate and update all cams in the list.



Preview of linear cam motion

Where do I find it?

Application	Modeling
Toolbar	**Die Validation® Linear Cam**
Menu	**Tools® Vehicle Manufacturing Automation® Die Validation® Linear Cam** **Linear Cams** group® **Preview Cam** **Linear Cams** group® **Recalculate Cam**
Location in dialog box	**Linear Cams** group® **Recalculate All Cams**

Rotary Cam

What is it?

You can now:

- Select the origin point of the cam axis.
- Recalculate the cam motion.
- Create multi-action cams.

Where do I find it?

Application	Modeling
Toolbar	**Die Validation® Rotary Cam**
Menu	**Tools® Vehicle Manufacturing Automation® Die Validation® Rotary Cam**
	Rotary Cams group® **Preview Cam**
	Rotary Cams group® **Recalculate Cam**
Location in dialog box	**Rotary Cams** group® **Recalculate All Cams**

Set Press Model

What is it?

You can use the new **Strokes Per Minute** option to set the press model timing information.

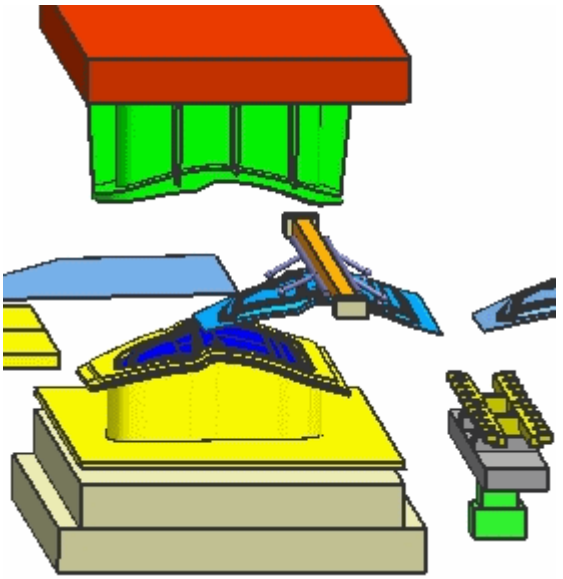
Where do I find it?

Application	Modeling
Toolbar	**Die Validation® Set Press Model**
Menu	**Tools® Vehicle Manufacturing Automation® Die Validation® Set Press Model**

Run Simulation

What is it?

You can now view the timing information during the simulation.



Where do I find it?

Application	Modeling
Toolbar	**Die Validation® Run Simulation**
Menu	**Tools® Vehicle Manufacturing Automation® Die Validation® Run Simulation**

Ship Design

Steel Features

Merged linear and non-linear steel features

What is it?

All linear and non-linear steel features are now merged into one feature, **Steel Features**.

Why should I use it?

This combined feature allows you to create on either a planar or non-planar face. Before this merger you had to know which face type you where going to use before you selected a steel feature. If you selected a planar steel feature you could not select a non-planar face. Now all you have to decide is what type of steel feature you want to create.

Where do I find it?

Application	Ship Design

Menu **Insert→Steel Features**

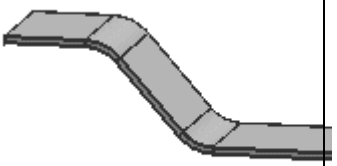
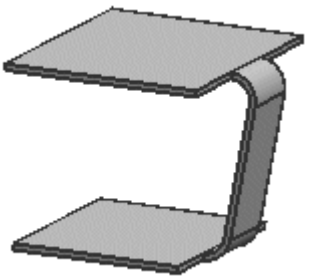
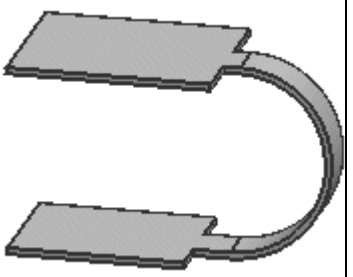
Flexible Printed Circuit Design

Bridge Transition

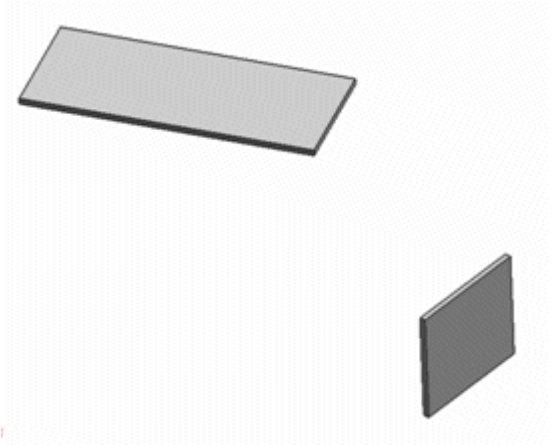
What is it?

Bridge Transition creates a shaped region that connects two distinct planar regions.

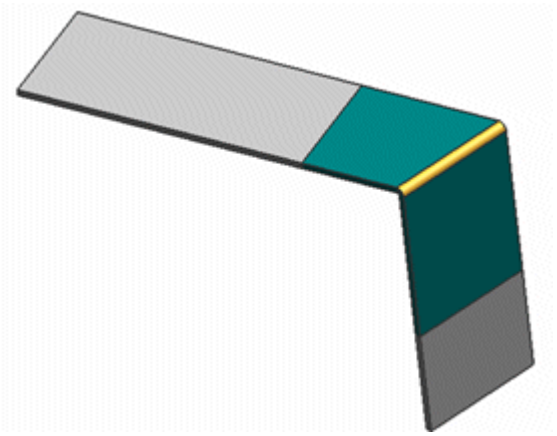
You can create a **Z**, **U**, or **Fold** shape, depending on the location and position of the geometry you want to connect.

Z	U	Fold
Consists of a planar region between two cylindrical bend regions. The axes of the bend regions are on opposite sides of the planar region. You can supply inner radii for the bends.	Consists of a planar region between two cylindrical bend regions. The axes of the bend regions are on the same side of the planar region. You can supply inner radii for the bends.	Consists of one cylindrical bend region between two planar regions. You supply a value to change the length of the planar region adjacent to the **Start Edge**.
		

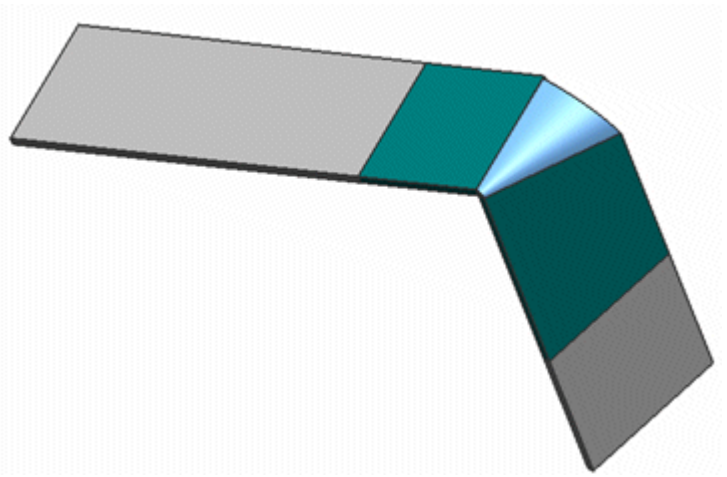
You can choose between cylindrical and conical bend regions when either option is possible, as for example, when the two planar regions you want to connect are not parallel.



Regions to be connected



Cylindrical transition



Conical transition

Why should I use it?

The Bridge Transition feature supports the typical workflow of flexible printed circuit design, which often begins with distinct planar regions and connects them with a transition region. This transition region can include bend as well as planar segments within it. This feature is an essential tool for designers of flexible printed circuits.

Where do I find it?

Application	Flexible Printed Circuit Design
Toolbar	**Flexible Printed Circuit Design**® **Bridge Transition**
Menu	**Insert**® **Flexible Printed Circuit Design Feature**® **Bridge Transition**
Location in dialog box	**Bend Parameters** group® **Allow Conical Bends** check box

PCB.xchange

Customizing work environment with site-specific settings

What is it?

You can now customize the PCB.xchange work environment with site-specific settings. To do this, you can set an environment variable to link to the network path where your default initialization files (.ini files) are stored. The environment variable MAYA_PCB_ENV_DIR can be set either by you on your local computer or by your network administrator.

Why should I use it?

If you use network search paths, you can set up a uniform work environment for people in your organization who collaborate on projects.

Commands available on the PCB.xchange toolbar and menu

What is it?

The following commands which were previously available only in the PCB.xchange window, are now available on the PCB.xchange toolbar and on the PCB.xchange menu:

- **Board Mesh and Thermal Settings**: Defines mesh controls and material and physical properties for the board.

- **Default Component Mesh and Thermal Settings**: Defines mesh controls and material and physical properties for the components.

- **Create ESC Solution**: Creates and exports NX Electronic Systems Cooling SIM and FEM files that are ready to be solved.

Why should I use it?

This improves the usability of the PCB.xchange toolbar.

Where do I find it?

Application	PCB.xchange For **Default Component Mesh and Thermal Settings**: A previously defined thermal database file containing material definitions and global component mesh collector properties
Prerequisite	For **Create ESC Solution**: A thermal component database file, Board Mesh and Thermal Settings and Default Component Mesh and Thermal Settings
Toolbar	**PCB.xchange® Board Mesh and Thermal Settings** **PCB.xchange® Default Component Mesh and Thermal Settings** **PCB.xchange® Create ESC Solution**
Menu	**PCB.xchange® Thermal / Flow Simulation® Board Mesh and Thermal Settings/ Default Component Mesh and Thermal Settings/ Create ESC Solution**

Board mesh simplification techniques

What is it?

Board simplification now uses the NX idealization process. Board simplifications can be revisited in the idealized part and modified at any time.

The process of removal of holes, blends, and other board features is simpler.

The idealized geometry automatically updates the NX generated board mesh in the *.fem* file.

Why should I use it?

The NX idealization process provides a more user-friendly approach to the board de-featuring and simplification tasks.

Where do I find it?

Application	PCB.xchange
Toolbar	**PCB.xchange® Board Mesh and Thermal Settings**
Menu	**PCB.xchange® Thermal / Flow Simulation® Board Mesh and Thermal Settings**

Junction-to-board coupling

What is it?

The new **Junction** option enables you to create a thermal coupling to simulate the junction-to-board coupling, when the most important pathway for the heat from a junction leaving a package is through the board.

Why should I use it?

Use this option to get a better estimate of the junction temperatures in components that do not have heat sinks attached to them.

Where do I find it?

Application	PCB.xchange
Prerequisite	A previously defined thermal database file containing material definitions and global component mesh collector properties
Toolbar	**PCB.xchange ® Default Component Mesh and Thermal Settings**
Menu	**PCB.xchange® Thermal / Flow Simulation® Default Component Mesh and Thermal Settings**
Location in dialog box	**Mesh Type**

Direct access to NX Electronic Systems Cooling from PCB.xchange

What is it?

You can now access the Advanced Simulation application from inside the PCB.xchange application. PCB.xchange automatically creates NX Electronic Systems Cooling SIM and FEM files of printed circuit assemblies.

To create the NX Electronic Systems Cooling files you must:

- Define a component thermal database XML file.
- Define **Board Mesh and Thermal Settings**.
- Define **Default Component Mesh and Thermal Settings**.

PCB.xchange automatically creates a FEM file with:

- Mesh collectors with the appropriate physical and mesh display properties in the *.fem* file.
- Material properties assigned to the mesh collectors for the components and the board.
- Meshes for the components and the board under the appropriate mesh collectors.

The SIM file is automatically associated with the FEM file previously defined. PCB.xchange automatically:

- Creates an active ESC solution in the *.sim* file.
- Applies the predefined thermal loads and boundary conditions to the components.
- Couples the component meshes with the board mesh using **Total Resistance** based **Thermal Couplings**.

The SIM file can then be setup to obtain the desired results types and solved.

Why should I use it?

The process of thermal simulation is automated for standard board designs.

Where do I find it?

Application	PCB.xchange
Prerequisite	A thermal component database file, **Board Mesh and Thermal Settings** and **Default Component Mesh and Thermal Settings**
Toolbar	**PCB.xchange** toolbar ® **Create ESC Solution**
Menu	**PCB.xchange**® **Thermal / Flow Simulation**® **Create ESC Solution**

New options to view restriction areas

What is it?

New options on the **PCB.xchange** menu let you control how you view the restriction area when you import or model a PCB.

- **Show Areas as Sketches** – Displays the restriction area as a sketch.
- **Show Area Heights** – Displays the restriction area as a transparent solid body.

You can switch between the two visual modes. You can set model translucency to display solids as opaque or transparent. For more information, see Setting Translucency.

You can also control the default display behavior of PCB.xchange by setting a variable in the initialization file *pcbx_ug_model.ini*.

Why should I use it?

Solid bodies provide visual feedback for interference inspections between components and restrictions areas. Sketches simplify display for models where height restrictions are less important.

Where do I find it?

Application	PCB.xchange
Prerequisite	An imported ECAD model or an NX CAD printed circuit assembly with restriction areas
Menu	**PCB.xchange® Tools® Show Areas as Sketches / Show Area Heights**

Chapter

10 Digital Simulation

Design Simulation

3D Tetrahedral Mesh enhancements

What is it?

This release includes a number of enhancements to the **3D Tetrahedral Mesh** capabilities.

3D Mesh dialog box changes

The **3D Mesh** dialog box has been redesigned to improve its overall usability and to make it consistent with NX user interface standards. For improved clarity, the names of several of the options in the **3D Mesh** dialog box have been modified:

Pre NX6 Option Name	NX 6 Option Name
Midnodes	Midnode Method
Surface Mesh Size Variation	Surface Curvature Based Size Variation
Volume Mesh Size Variation	Element Growth Rate Through Volume
Attempt Mapping	Attempt Free Mapped Meshing
Mesh Transition	Transition Element Size

Automatic Element Size now available with multiple bodies

You can now use the **Automatic Element Size** option even if you selected several bodies to mesh. In previous releases, if you selected more than one body, the **Automatic Element Size** option was unavailable.

Improvements for meshing multiple bodies

When you use **3D Tetrahedral Mesh** to mesh multiple bodies simultaneously, the software now uses logic to determine which bodies to mesh first. The software now meshes smaller bodies and bodies that have existing constraints first. This can prevent conflicts which can occur.

Model cleanup process improvements

This release includes improvements to how the software performs the automatic model cleanup operations that occur during the meshing process. When you select multiple bodies with **3D Tetrahedral Mesh**, the software now performs all the cleanup (abstraction) operations on all the bodies before it begins meshing.

Where do I find it?

Application	Design Simulation
Toolbar	**Advanced Simulation® 3D Tetrahedral Mesh**
Menu	**Insert® Mesh® 3D Tetrahedral Mesh**

New customer default for resetting model cleanup operations

What is it?

A new **Automatically Reset Geometry** option has been added to the General page of the **Meshing** customer defaults. Use this option to control how the software handles the deletion of any automatic or manual model cleanup operations (geometry abstractions) if you sufficiently reduce either the **Small Feature Tolerance** or **Element Size** for an existing mesh.

- If you select **Never**, the software preserves all abstractions. Any reduction you make in the **Small Feature Tolerance** or **Element Size** has no effect.

- If you select **Always**, the software deletes all existing abstractions, including any manual abstractions that you created with the commands on the **Model Cleanup** toolbar.

- If you select **Ask**, the software prompts you for confirmation before it deletes any abstractions.

Where do I find it?

Application	Design Simulation
Menu	**File® Utilities® Customer Defaults®**
Location in dialog box	**Simulation® Meshing®** General page

Checking node proximity to underlying geometry

What is it?

Use the new **Node Proximity to CAD Geometry** option in the **Model Check** dialog box to check the proximity of the nodes to the original CAD geometry. This lets you evaluate the fidelity of the mesh (which the software creates on the polygon geometry) to the underlying CAD part.

In the **Model Check** dialog box, you specify a **Proximity Tolerance** to define the maximum distance a node can lie from the corresponding CAD edge or face. When you click **OK** or **Apply**, the software evaluates the nodes and reports any that exceed the tolerance. If the software finds such nodes, you can use the **Adjust Node Proximity** option to move those nodes so they lie within the tolerance.

Note

You must have the idealized part loaded to use the **Node Proximity to CAD Geometry** option.

Why should I use it?

The **Node Proximity to CAD Geometry** check is most useful before you perform a contact analysis when you need to ensure the proximity of node locations in regions of contact. Prior to meshing the part, the software performs a number of automatic cleanup operations. These cleanup operations, for example, remove sliver surfaces and create seams in periodic faces. The software then generates the mesh. However, as a result of these cleanup operations, the nodes may not always be located exactly on the CAD surfaces. While small differences in node position are inconsequential for most types of analyses, they can be significant for the accurate detection of contact.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains a mesh and the idealized part loaded
Toolbar	**Advanced Simulation® Finite Element Model Check**
Menu	**Analysis® Finite Element Model Check**

Create and Edit Solution usability improvements

What is it?

This release includes improvements to the Nastran **Create Solution** and **Edit Solution** dialog boxes.

Create and Edit Solution dialog boxes reorganized

The options in the **Create Solution** and **Edit Solution** dialog boxes have been reorganized to make their correspondence to the different sections of the Nastran input file more clear. The options are now organized according to the section of the Nastran input file to which they belong. For example, the **Geometry Check** and **Max Job Time** options appear on the Executive Control page of options. This new organization more closely mirrors the structure of an actual Nastran input file and makes options easier to find.

Ability to preview a Nastran input file

Use the new **Preview Solution Setup** option in the **Create Solution** and **Edit Solution** dialog boxes to preview the associated Nastran solver input file in an information window. When you click **Preview**, the software displays the syntax of the input file based on the selected solution options. You can use the previewed file to validate that you have correctly specified all the necessary options for your analysis.

Note

The previewed input file is an abbreviated version of the actual input file. The software omits all node, element, load, and constraint data from the bulk data section.

Where do I find it?

Application	Design Simulation
Simulation Navigator	Right-click the Simulation® **New Solution**, or right-click the current solution® **Edit Solution**
Menu	**Insert**® **Solution**

Materials enhancements

What is it?

The process for working with materials has been changed so that the most common task of assigning a material is easier. You no longer have to work through a series of additional dialog boxes to select and assign the material.

Material Properties opens the **Assign Material** dialog box, where you can assign an existing material to a model and perform additional tasks, such as creating a new material.

Where do I find it?

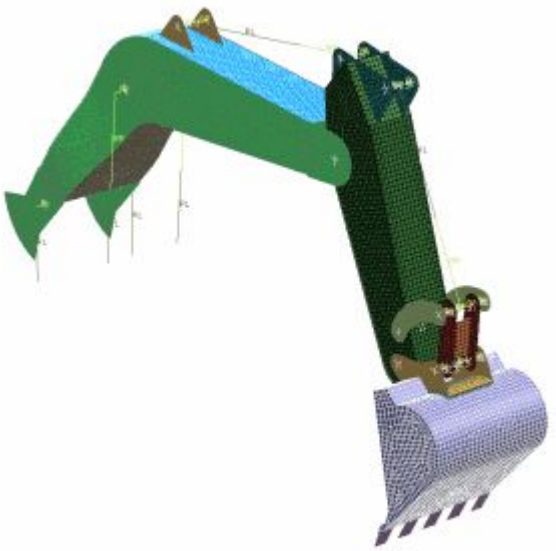
Application	Design Simulation
Prerequisite	FEM, idealized part, or part active
Toolbar	**Design Simulation® Material Properties**

Advanced Simulation

Assembly FEM

What is it?

An assembly FEM (.afm) is a new simulation file type that supports enhanced workflows for analyzing large assemblies. Assembly FEMs are similar to part assemblies. Much like the way a part assembly contains occurrence and position data for multiple component parts, an assembly FEM contains occurrence and position data for multiple component FEMs. In addition, the assembly FEM contains the connection elements (such as spider elements, weld elements, and so on) that join component FEMs into a system.



Mapping component FEMs to assembly part instances

Assembly FEMs support two basic workflows:

- Associative. In this workflow, you associate an assembly FEM with an existing assembly of parts, and map new or existing component FEMs to each component part. When the assembly configuration or the geometry of its component parts is updated, the assembly FEM is also updated.

  You can combine associative and non-associative component FEMs within the same assembly FEM.

- Non-associative. In this workflow, you first create an empty assembly FEM. You then add component FEMs to the assembly FEM. Finally, you use **Reposition Component** to define the position and orientation of component FEMs.

Assembly FEMs support multiple FEM occurrences and subassembly FEMs. That is, you can map the same FEM to multiple occurrences of a part in the assembly hierarchy, and you can map an assembly FEM to a subassembly within a larger assembly FEM.

Edits to component FEMs are immediately reflected in the assembly FEM, and in all occurrences of the component FEM.

The **Assembly Label Manager** provides tools to resolve node, element, and coordinate system labeling conflicts.

Support for Assembly FEM management is included in Teamcenter Integration using the Teamcenter for Simulation data model.

Why should I use it?

In previous releases, the only way to model an assembly was to create a single FEM for the assembly part and mesh each component part as a polygon body in the FEM. While this approach is still available, and may be appropriate for simple, small assemblies, assembly FEMs provide the following advantages when working with larger, more complex assemblies:

- For large models consisting of FE data with no underlying geometry, assembly FEMs provide improved documentation and management of component meshes.

- You can use and reuse existing component FEMs, including legacy and imported FEM data, in multiple assembly FEMs.

- You can control the loading of component FEMs for more efficient use of resources.

- You can replace individual component FEMs with alternate mesh or geometry representations, to support what-if analyses while retaining the original component FEM data and conserving effort and resources.

- Assembly FEMs support distributed workflows. Team members or third parties provide meshes for individual parts or subassemblies, which an analyst or project leader can assemble into a full system model. Updates to component FEMs or their associated CAD data can be handled automatically by the software, or user-controlled on a part-by-part basis.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Standard**® **New**
Menu	**File**® **New**
Simulation Navigator	**Simulation File View**® right-click a part assembly file® **New Assembly FEM**
Location in dialog box	**New** dialog box® **Simulation** tab® **Templates** group® **Blank – Assembly FEM** template

Data management

Automatically load associated parts

What is it?

You can now specify that the software load all associated data automatically when you open a FEM or Simulation file, using the new **Automatically Load Associated Parts for Component FEM** customer default.

In previous releases, when you opened an existing FEM or Simulation file, the associated part and idealized part files were not loaded at the same time.

Why should I use it?

When working with large models, not loading associated parts can save system resources. However, certain analyses, such as Optimization, require that the idealized part file is loaded. In addition, automatically loading associated parts can be convenient if you will be performing additional geometry idealization or modifying part geometry.

Where do I find it?

Menu	**File**® **Utilities**® **Customer Defaults** **Simulation**® **General**® Environment
Location in dialog box	page® **Automatically Load Associated Parts for Component FEM** check box.

Teamcenter Integration enhancements

What is it?

There are several enhancements to Teamcenter Integration when managing simulation data in NX using the Teamcenter for Simulation data model, including support for the following:

- CAE Structure Editor
- Custom subtypes
- Reports

CAE Structure Editor support

Any changes you make to the CAE data relationships (idealized part, FEM, and Simulation item revisions) using the CAE Structure Editor in the Teamcenter client are reflected when you open those item revisions in NX. In addition, you can create empty CAE data item revisions in the CAE Structure Editor, and open those items in NX. The corresponding datasets are created in Teamcenter Integration when you save your work.

Custom subtypes support

If your Teamcenter installation defines custom subtypes for CAE items, you can specify the subtype when creating or migrating data in NX.

Reports support

You can store and manage HTML reports created for solved models in NX as named references in the CAEAnalysis dataset. When you save a solved model, you are prompted to import the HTML report and all referenced files, in addition to any solver-generated files.

Where do I find it?

Application	Advanced Simulation
Prerequisites	Teamcenter Integration for NX, Teamcenter for Simulation In the Teamcenter Integration customer defaults, **Structure Update on Load** must be set to **Complete**, and **Structure Update on Save** must be selected.

General capabilities

Smart selection enhancements

What is it?

Smart selection methods enable you to select entities according to specified criteria or entity relations. For this release, support for smart selection methods has been extended to many additional commands, including:

- Meshing
- Manual node and element operations
- Groups

When selecting entities, you can use the **Method** list on the Selection bar to select related entities.

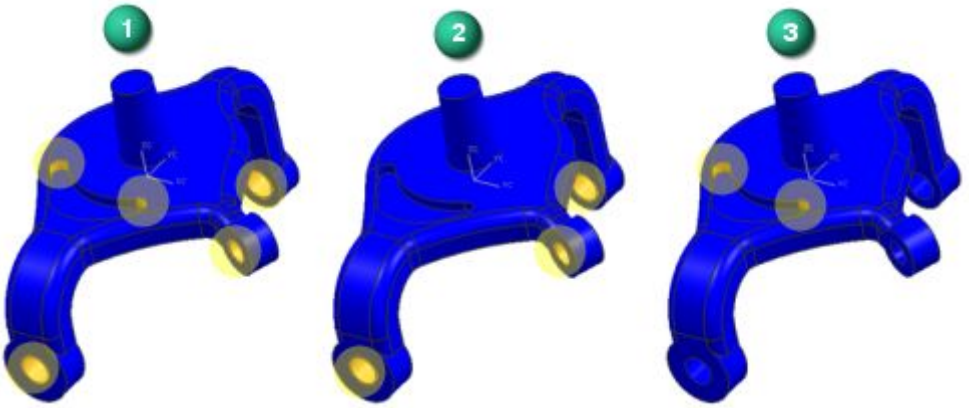
The Selection Bar, showing (1) the Type Filter list, (2) the Method list, and (3) the Smart Selector Options button

This release also introduces several new options for controlling the selection of CAE items using the smart selection methods. These include:

- For all selection methods, you can specify whether only visible entities are selected (the default), or all entities regardless of visibility.
- The **Cylinder Faces** method now supports the selection of two half-cylinders, four quarter-cylinders, and so on. You can also specify a minimum and maximum cylinder angle.
- The **Fillet Faces** method now supports the selection of inside fillets, outside fillets, or both. You can also specify a minimum and maximum fillet angle.

In addition, the **Smart Selector Options** dialog box now displays only the options appropriate for the current selection method.

By specifying the minimum and maximum angle for cylinder and fillet faces, you can more effectively filter the selection to select only the faces you want, as demonstrated in the following figure.



1
2
3

The minimum angle is set to 150, and the maximum angle is set to 360. The software selects the cylindrical holes as well as two partial cylindrical faces.
The minimum angle is set to 200, and the maximum angle is set to 360. The software selects only the cylindrical holes.
The minimum angle is set to 150, and the maximum angle is set to 200. The software selects only the two partial cylindrical faces.

Why should I use it?

Use the selection methods whenever you need criterion-based selection of polygon geometry, nodes, elements, and other FE entities.

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.

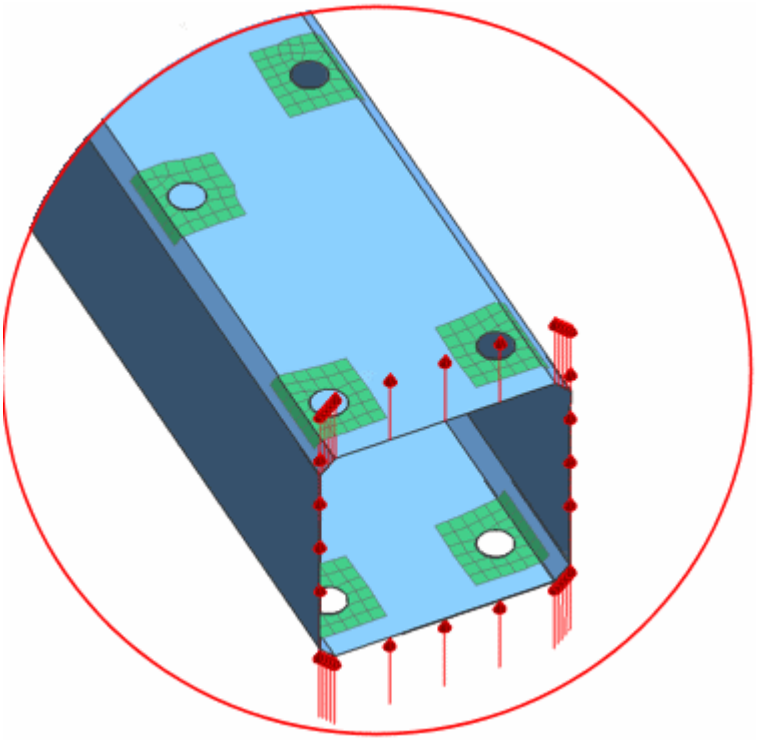
Where do I find it?

Application	Advanced Simulation
Prerequisite	In a command that supports smart selection, choose a selection method in the **Selection Bar**.
Toolbar	**Selection Bar® Smart Selector Options**

FE Groups

What is it?

You can now organize your model into subsets using groups. Groups are user- or system-defined collections of FE and/or design entities.



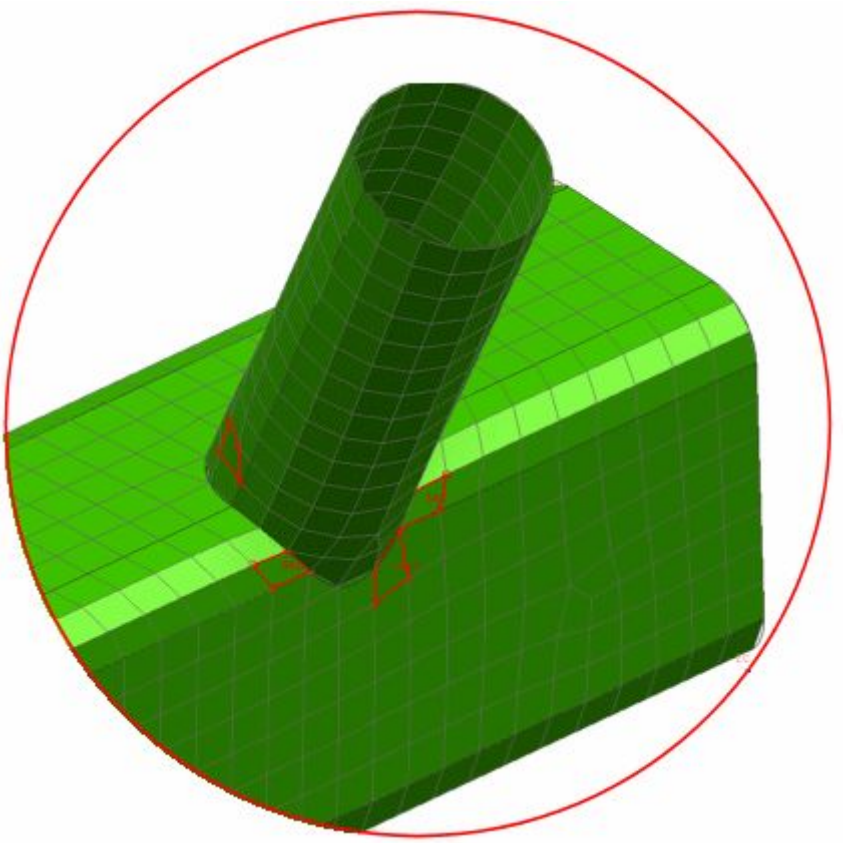
You can store the following entities in a group:

- Nodes
- Elements
- Meshes
- Mesh points
- Points
- Polygon faces
- Polygon bodies
- Curves
- Coordinate systems

Why should I use it?

Grouping has several advantages. You can:

- Reduce model complexity by separating components of your model into individual groups to display individually or more than one at a time.
- Select a smaller portion of the model for the application of a load or other boundary condition.
- Store elements that failed a quality check.



For more information, see Groups support in Post-processing.

Where do I find it?

Application	Advanced Simulation
Menu	**Format→Group→New Group** Right-click the Simulation file node/FEM file node® **New Group**
Simulation Navigator	Right-click an existing group® **New Group / Manage / Sort Alphabetically**
Location in dialog box	**Group Manager** dialog box® **General** group

Field enhancements

What is it?

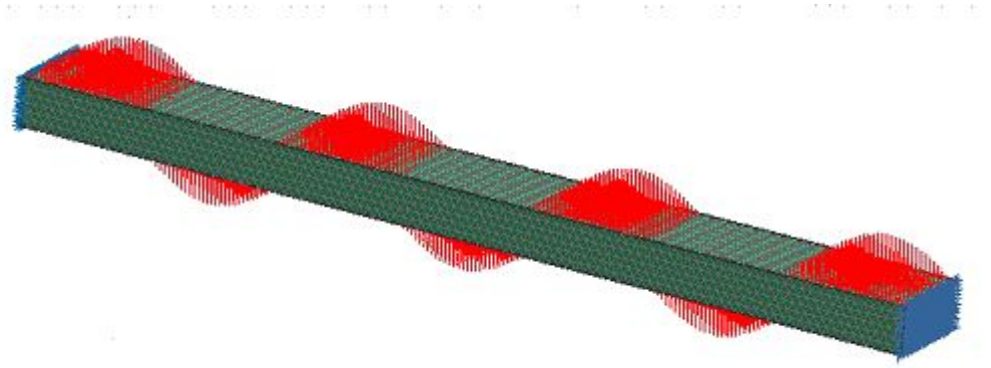
Fields are used to define variant functions that specify magnitudes or properties in terms of other variables such as time.

Fields have the following enhancements:

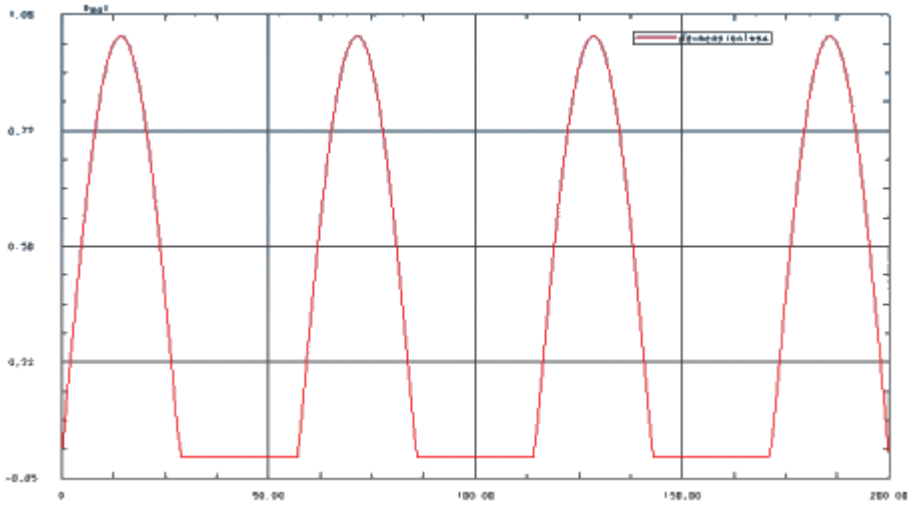
- The user interface has been redesigned to improve usability and consistency. You can choose to define a field by creating a formula, table, or by linking to an existing field.

- Fields can be displayed as XY plots.

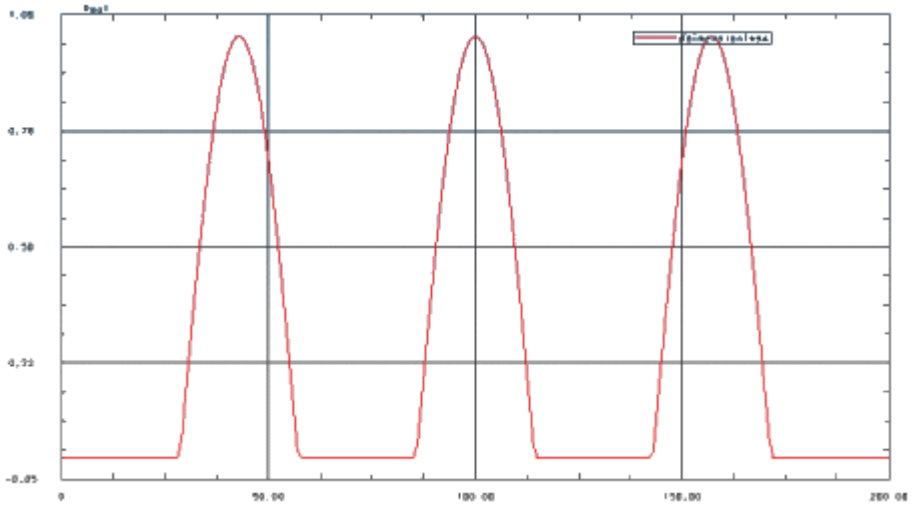
  The figure shows two normal forces. The force on the top of the model is distributed spatially using a field, as is the force on the bottom of the model.



  The spatial distribution of the force on the top of the model is shown in the following XY plot.



  The spatial distribution on the bottom of the model is shown below.



- Tables are easier to edit. You can also import a table from a file, and edit a table in a spreadsheet application.

- You can now manage fields using the **Part Navigator**. You can plot, edit, export, copy, rename, or delete a field from the **Part Navigator**. The **Fields** dialog box, which was used to manage fields, has been removed.

- For appropriate boundary conditions, you can use a spatial independent domain.

Why should I use it?

You can now use fields to define material properties, and to define how the magnitude of a boundary condition is distributed spatially.

Where do I find it?

Application	Advanced Simulation **Insert® Field** **Insert® Predefined Formula Field** **Insert® Predefined Table Field**
Menu **Part Navigator**	**Insert® Import Field** right-click the **User Fields** node
Location in loads and constraints dialog boxes	**Specify Field** list This option also appears in the **Hyperelastic Materials** dialog box.

Geometry idealization

Split Body command available in Advanced Simulation

What is it?

When you are working with an idealized part, the **Split Body** command from Modeling is now available in Advanced Simulation. **Split Body** has been significantly enhanced in this release, and it now provides a more robust and flexible alternative to the **Partition** command when you are preparing geometry for analysis.

Use **Split Body** to divide the target geometry into one or more bodies. You can use **Split Body** to divide either sheet or solid bodies. **Split Body** produces an associative feature that appears in the model's history that you can update, edit, or delete as necessary.

With **Split Body**, you use a "tool" body or geometry to divide the target geometry in the locations you specify. With **Split Body**, that tool geometry can be:

- An existing face or datum plane.
- A new datum plane.
- A revolved or extruded body that you can create directly from the **Split Body** dialog box.

  Note

  You must first promote the bodies you want to split before you use the **Split Body** command. To promote a body in Advanced Simulation, highlight the idealized part in the **Simulation Navigator** and select **Promote** from the right-click menu.

Mesh mating conditions must be created manually

Unlike the **Partition** command, **Split Body** does not currently produce mesh mating conditions at the locations where you divide the target geometry. After you use **Split Body** in the idealized part, and you make the FEM file active, you must use the **Mesh Mating Condition** command to create connections between the divided bodies. This ensures that the meshes on the split bodies are continuous.

Why should I use it?

Use **Split Body** to help prepare complex geometry for meshing. For example, you can use **Split Body** to subdivide a larger model into smaller, sweepable regions to facilitate hexahedral meshing.

Where do I find it?

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

Material and physical properties

Materials enhancements

What is it?

Advanced Simulation now supports a nonlinear hyperelastic material type that is used by the NX Nastran nonlinear solver. This general hyperelastic material corresponds to the MATHP bulk data entry in NX Nastran and MSC Nastran.

Also in this release, the process for working with materials has been changed so that the most common task of assigning a material is easier. You no longer have to work through a series of additional dialog boxes to select and assign the material.

Material Properties opens the **Assign Material** dialog box, where you can assign an existing material to a model and perform additional tasks, such as creating a new material. The **Assign Material** dialog box also opens when you click **Choose Material** on a physical properties dialog box.

Where do I find it?

Application	Advanced Simulation
Prerequisite	FEM, idealized part, or part active
Toolbar	**Advanced Simulation® Material Properties**
Location in physical property creation dialog box	**Choose Material**

Material orientation support

What is it?

This release includes support for defining material orientation vectors for selected elements. For certain types of Nastran and ANSYS shell elements, you can use the new **Element Associated Data** capability to specify material orientation vectors. The ability to define material orientation vectors is necessary, for example, in laminate modeling where you need to individually

orient elements in each layer. See Ability to define properties on individual elements for more information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains elements for which material orientation is supported.
Toolbar	Advanced Simulation® **Element Associated Data**
Menu	**Edit® Element® Modify Associated Data**

Attribute Editor is now Mesh Associated Data

What is it?

In previous releases, you used the **Attribute Editor** dialog box to specify the properties for the elements in a mesh. You could only define those properties by editing an existing mesh; there was no way to specify the properties at the time you initially generated the mesh.

In this release, the **Attribute Editor** dialog box has been renamed **Mesh Associated Data** to clarify its purpose.

Mesh Associated Data available from meshing dialog boxes

To improve the workflow for defining these properties, you can now define **Mesh Associated Data** directly from the meshing dialog boxes. If you select an element type that requires the definition of additional properties, a **Mesh Associated Data** button appears. This button:

- Provides you with a visual clue that additional properties are required for an element type.
- Allows you to define those properties as you are initially creating the mesh.

Comparing Mesh Associated Data and Element Associated Data

With **Mesh Associated Data**, the software applies the properties uniformly to all the elements of that type in the associated mesh collector. This is different from **Element Associated Data**, introduced in this release, which you can use to specify different values for a subset of properties to individual elements. This allows you to vary properties, such as material orientation, on an element by element basis. See Ability to define properties on individual elements for more information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains elements for which additional properties are required.
Menu	**Tools® Mesh Associated Data**
Simulation Navigator	Right-click the appropriate mesh® **Edit Mesh Associated Data**

Ability to define properties on individual elements

What is it?

In this release, you can use the new **Element Associated Data** command to define certain properties for individual elements. Depending upon your solver, this allows you to define properties, such as mass or material orientation, that vary across different elements. In previous releases, you could only specify a uniform set of properties across an entire mesh.

Solver	Element type	Supported associated data
Nastran	CQUAD4	Material orientation (by vector projection or coordinate system)
	CQUAD8	
	CQUADR	Corner node thickness. This shell thickness overrides the thickness defined in the mesh.
	CTRIA3	
	CTRIA6	
	CONM2	Mass
	CBAR, CBEAM	Orientation of the cross section on the beam element. With 1D meshes, the element-associated data always overrides the mesh-associated data.
	CELAS2	Spring stiffness (translational or rotational)

Solver	Element type	Supported associated data
ANSYS	SHELL63(4) and (3) SHELL93(8) and (6) SHELL91(8) and (6) SHELL99(8) and (6) SHELL181(4) and (3)	Material orientation Shell thickness. This shell thickness overrides the thickness defined in the physical property table. This value is stored in the R or RMORE card in the ANSYS input file. **Note** If you import an ANSYS input file into NX, R cards will be translated to the physical property table rather than element-associated data.
LS-DYNA	ELEMENT_SHELL(3) ELEMENT_SHELL(4) ELEMENT_SHELL(6) ELEMENT_SHELL(8)	Material orientation **Note** With LS-DYNA, you can define element attributes at either the element level or the mesh level. If you select **Use Element Associated Data** for the mesh, and not all elements have material orientations defined, those elements are written to the keyword file as *ELEMENT_SHELL. This is the same as selecting **None**

Solver	Element type	Supported associated data
		for the **Keyword Option**.

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

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**

Meshing

General meshing enhancements

What is it?

This release includes a number of enhancements to the meshing capabilities within Advanced Simulation.

Meshing dialog boxes redesigned

The following meshing dialog boxes have been redesigned to improve their overall usability and make them consistent with NX user interface standards:

- **0D Mesh**
- **1D Mesh**
- **2D Mesh**
- **3D Tetrahedral Mesh**
- **3D Swept Mesh**

Support for smart selection

You can now use the meshing commands in conjunction with the Advanced Simulation smart selection methods. These methods make it easier to select specific types of geometry, such as fillets, when you create a mesh. See Smart selection enhancements for more information.

Convenient access to Mesh Associated Data dialog box

When you select an element type which requires that you define additional properties, a new **Edit Mesh Associated Data** button appears in the **Element Properties** group. Click **Edit Mesh Associated Data** to open the **Mesh Associated Data** dialog box (called the **Attribute Editor** dialog box in previous releases) where you can specify those properties. For example, if you are working with the NX Nastran solver and select CQUAD4 from the element **Type** list in the **2D Mesh** dialog box, you can click **Edit Mesh Associated Data** to define an offset value for all the elements in the mesh. In previous releases, you could not define these types of properties when you initially created a mesh.

Problematic geometry for meshing placed in an output group

Beginning in this release, if the software encounters geometry that it is unable to mesh, it places that geometry in an output group in the Simulation Navigator. You can then use the output group to easily display, examine, and repair any geometry issues with that geometry. For more information on the new grouping capabilities introduced in this release, see FE Groups.

Journaling support

The meshing commands now support the NX journaling capabilities. Journaling is a rapid automation tool that you can use to record, edit, and replay interactive NX sessions. You can create journals, for example, to automate repetitive tasks or workflows. For more information, see Journaling Overview in the NX Help.

Newly supported element types

What is it?

In this release, support has been added for a number of new element types for the NX Nastran, ABAQUS, and ANSYS solvers.

New NX Nastran Element Types

Element Name	Description
CQUADX4	Axisymmetric linear quadrilateral element
CQUADX8	Axisymmetric parabolic quadrilateral element
CTRAX3	Axisymmetric linear triangular element
CTRAX6	Axisymmetric parabolic triangular element

New ABAQUS Element Types

Element Name	Description
B21	2-node linear beam element in a plane

B21H	2-node linear beam element in a plane, hybrid formulation
B31H	2-node linear beam element in space, hybrid formulation
S4R5	4-node doubly curved thin shell element with reduced integration, hourglass control, using 5 degrees-of-freedom per node
STRI3	3-node triangular facet thin shell element

New ANSYS Element Types

Element Name	Description
BEAM188	3D linear finite strain beam element
LINK8	3D spar or truss element
LINK10	3D tension-only or compression-only spar element
INTER192	2D 4-node gasket element
INTER195	3D 8-node gasket element

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with the appropriate solver specified
Toolbar	**Advanced Simulation** toolbar® **1D Mesh** , **2D Mesh** , **3D Mesh**
Menu	**Insert→Mesh→1D Mesh**, **2D Mesh**, **3D Mesh**

Expanded pyramid element support

What is it?

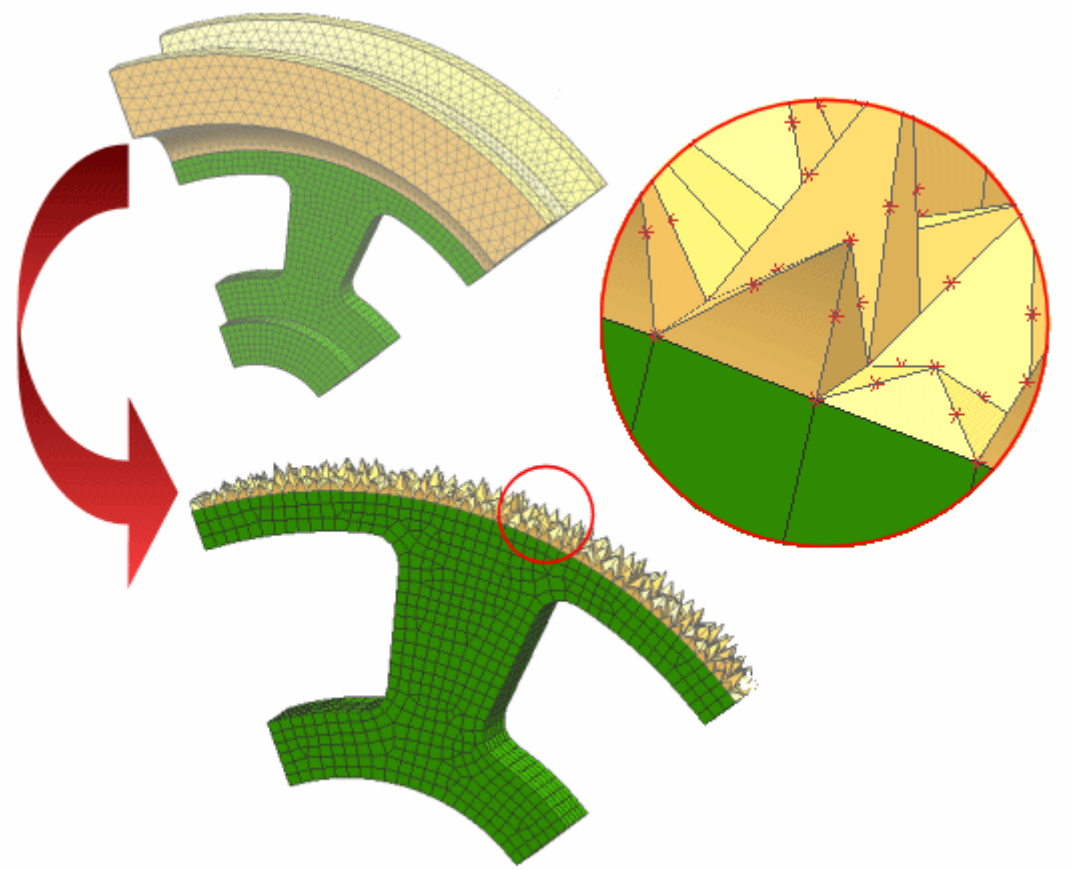
This release includes improved support for pyramid elements. If you are working in the ANSYS solver language, when you generate a tetrahedral mesh on a body (volume) that is adjacent to a body with an existing hexahedral mesh, you can select the **Use Pyramids for Transition** option in the **3D Tetrahedral Mesh** dialog box to create transitional pyramid elements between the meshes.

Ability to create pyramid transitions between elements of different orders

You can now use pyramid elements to transition between hexahedral and tetrahedral meshes of different orders. For example, you can use pyramid elements to transition from a mesh of linear hexahedral elements to a mesh of parabolic tetrahedral elements. Previously, you could only use pyramid elements to transition between meshes of the same order.

Support for pyramid and tetrahedral elements with missing midside nodes

To support pyramid transitions between elements of different orders, Advanced Simulation now supports “mixed order” pyramid and tetrahedral elements. These elements contain a mix of linear and parabolic edges; the parabolic edges have midside nodes, while the linear edges do not. With a mixed order pyramid element, typically the edges of its base are linear while the other edges are parabolic. However, it is possible to have pyramid and tetrahedral elements with any number of missing midside nodes.



Note

You cannot use the manual element creation commands on the **Element Operations** toolbar with pyramid or tetrahedral elements that have missing midside nodes.

For more information, see Pyramid element transitions in the Advanced Simulation help.

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 ANSYS as the specified solver
Toolbar	**Advanced Simulation® 3D Tetrahedral Mesh**
Menu	**Insert® Mesh® 3D Tetrahedral Mesh**

1D, 2D, and 3D meshes

1D Mesh process changes

What is it?

As part of the usability redesign of the **1D Mesh** dialog box, several operations that you previously performed from the **1D Mesh** dialog box have been relocated.

- The **Create Weld Elements** option has been replaced by the new **Spot Weld** command. See Spot Weld for more information.
- In previous releases, you could use the **1D Mesh** dialog box to create a mesh between selected edges. In this release, that capability has been moved to the new **1D Connection** dialog box. See 1D Connection for more information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 1D Mesh**
Menu	**Insert® Mesh® 1D Mesh**

2D Mesh enhancements

What is it?

This release includes a number of enhancements to the **2D Mesh** capabilities.

2D Mesh dialog box changes

The **2D Mesh** dialog box has been redesigned to improve its overall usability and to make it consistent with current NX user interface standards.

- Options that were located in the **Mesh Options** dialog box in previous releases are now available directly in the **2D Mesh** dialog box.

- For improved clarity, the names of several of the options in the **2D Mesh** dialog box have been modified:

Pre-NX6 Option Name	NX 6 Option Name
Midnodes	Midnode Method
Mesh Size Variation	Curvature Based Size Variation
Attempt Mapping	Attempt Free Mapped Meshing
Mesh Transition	Transition Element Size
Number of Elements per Quarter Round	Number of Elements per 90 deg

Improved handling for fillets and cylinders

In previous releases, you used the **Fillet Pre-Processing** options in the **Mesh Options** dialog box to control how the software generated the mesh along fillets and, to a limited extent, cylinders. In this release, you now use the new **Fillet Faces** and **Cylinder Faces** smart selection methods in conjunction with the **2D Mesh** dialog box to control mesh generation on fillets and cylinders as follows:

1. Click **2D Mesh** .

2. In the **Selection Bar**, select either **Fillet Faces** or **Cylinder Faces** from the smart selection method list.

3. Click **Smart Selector Options** and use the options in the **Smart Selector Options** dialog box to specify the criteria for the fillets or cylinders.

4. In the **2D Mesh** dialog box, click **Select Objects** and select the appropriate geometry.

If the software detects any fillets or cylinders that meet the specified criteria, it displays additional options in the **2D Mesh** dialog box:

- With fillets, you can use the **Elements per 90 deg** option to control the number of elements the software generates along each 90° segment of the fillet.

- With cylinders, you can use the **Elements per 90 deg** option to control the number of elements the software generates along each 90° segment of the cylinder. You can also use the **Element Size along Cylinder Height** to control the size of the elements generated along the cylinder's axis.

In previous releases, you had no way to specify the size of the elements along the cylinder's height.

See Smart selection enhancements for more information on the new selection methods.

Ability to easily add or remove faces from an existing mesh

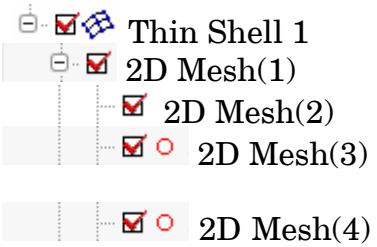
When you edit a mesh, you can now easily add or remove faces from that mesh. In previous releases, there was no easy way to remove a face from an existing mesh. You first had to delete the entire mesh and then remesh the remaining faces without the unwanted face. Now, when you right-click on an existing mesh in the graphics window and select **Edit**, you can use the **Select Objects** option in the **2D Mesh** dialog box to select the unwanted face. When you click **OK**, the software removes that face from the mesh.

Support for partial mesh results

In previous releases, if the software was unable to generate a mesh on one or more selected faces, it did not produce a mesh on any of the selected faces. Beginning in this release, the software now places any failing faces in an output group in the Simulation Navigator and generates a mesh on the remaining faces. You can then use the output group to easily display, examine, and repair any geometry issues with those faces. For more information, on the new grouping capabilities introduced in this release, see FE Groups.

New Simulation Navigator indicator for Export Mesh to Solver status

For 2D meshes for which you cleared the **Export Mesh to Solver** option, the status column in the **Simulation Navigator** now displays that the mesh is "Not exported to solver." Additionally, a graphical indicator (○) now appears adjacent to such meshes, such as 2D Mesh (3) and 2D Mesh (4) shown below:



These changes help you quickly identify which 2D meshes will not be exported to the solver input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file

Toolbar

Menu **Advanced Simulation® 2D Mesh**
Insert® Mesh® 2D Mesh

3D Tetrahedral Mesh enhancements

What is it?

This release includes a number of enhancements to the **3D Tetrahedral Mesh** capabilities.

3D Mesh dialog box changes

The **3D Mesh** dialog box has been redesigned to improve its overall usability and to make it consistent with NX user interface standards. For improved clarity, the names of several of the options in the **3D Mesh** dialog box have been modified:

Pre NX6 Option Name	NX 6 Option Name
Midnodes	**Midnode Method**
Surface Mesh Size Variation	**Surface Curvature Based Size Variation**
Volume Mesh Size Variation	**Element Growth Rate Through Volume**
Attempt Mapping	**Attempt Free Mapped Meshing**
Mesh Transition	**Transition Element Size**

Fillet and cylinder options now defined through 2D Mesh dialog box

In previous releases, you could define options for meshing fillet faces through the **3D Tetrahedral Mesh** dialog box. These options controlled the 2D mesh the software generated on those faces. The software then used this 2D mesh to seed the 3D mesh.

As part of the redesign of the meshing dialog boxes, these options have been removed from the **3D Tetrahedral Mesh** dialog box. Now, you first create surface meshes on fillets and cylinders through the **2D Mesh** dialog box. You can then use the **3D Mesh** dialog box to define the mesh on the volume. See 2D Mesh enhancements for more information.

Option to apply Automatic Element Size to multiple bodies

You can now use the **Automatic Element Size** option even if you selected several bodies to mesh. In previous releases, if you selected more than one body, the **Automatic Element Size** option was unavailable.

Improvements for meshing multiple bodies

When you use **3D Tetrahedral Mesh** to mesh multiple bodies simultaneously, the software first evaluates which body it should mesh first. The software now meshes smaller bodies and bodies that have existing constraints first. This can prevent possible conflicts.

Model cleanup process improvements

This release includes improvements to the automatic model cleanup operations the software performs during the meshing process. When you select multiple bodies with **3D Tetrahedral Mesh**, the software now performs all the cleanup (abstraction) operations on all the bodies before it begins meshing.

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**

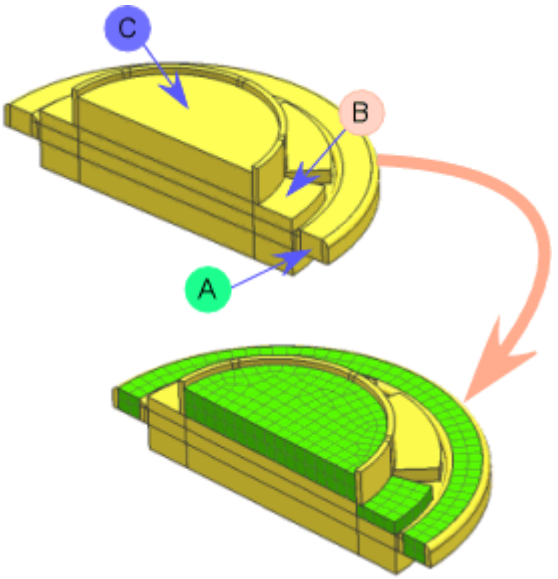
3D swept meshes on multiple bodies

What is it?

You can now use **3D Swept Mesh** to generate hexahedral meshes on multiple bodies simultaneously. Use the options in the new **Type** menu to control how the software sweeps the mesh through the selected bodies.

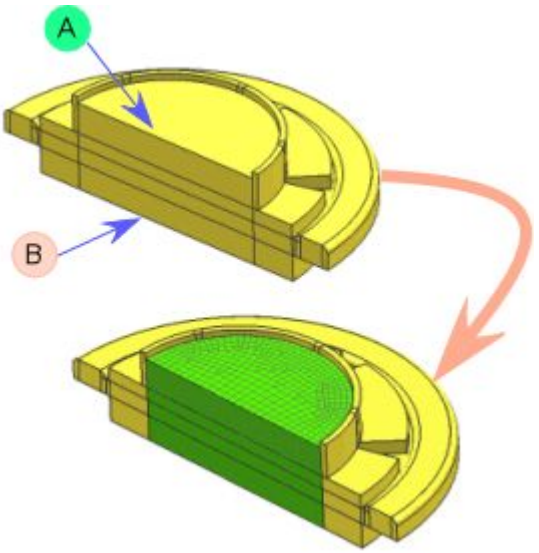
- Select **Multi Source** to sweep a mesh from a selected source face to a target face determined by the software. With this option, the software sweeps the mesh through each individual body. However, you can select source faces in separate bodies simultaneously.

  In this example, we selected **Multi Source** from the **Type** list. We then selected faces A, B, and C as the source faces. Notice that the source faces each belong to a different body. When we clicked **OK**, the software swept a mesh through each of the three bodies.



- Select **Until Target** to sweep a mesh from a source face in one body to a target face in another body, as long as the intervening bodies are connected to each other with mesh mating conditions.

  In this example, we selected **Until Target** from the **Type** list. Then selected face A as the source face and face B (on the bottom of the model) as the target face. When we clicked **OK**, the software swept the mesh from the source face, through the intermediate bodies, to the target face.



In previous releases, you could only sweep a mesh through a single solid body at a time.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Advanced Simulation® 3D Swept Mesh**
Menu	**Insert® Mesh® 3D Swept Mesh**

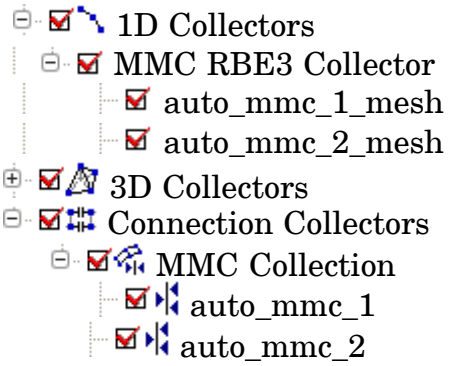
Connection meshes

Mesh Mating Conditions enhancements

What is it?

When you define a glue non-coincident mesh mating condition, and Nastran is the selected solver, the software now defines it as a 1D mesh of RBE3 elements. This 1D mesh is generated when the mated faces are meshed.

The generated mesh appears in the **Simulation Navigator** under the **1D Collectors** node. You can manage and edit this mesh like any other mesh. You can control the display and appearance of the mesh. You can right-click the mesh and select **Edit Mesh Associated Data** to edit core and leg node degrees of freedom for the RBE3 elements in the mesh.



When you create glue non-coincident mesh mating conditions (A), the software generates corresponding RBE3 meshes (B).

Why should I use it?

In previous releases, the generation of glue non-coincident mesh mating conditions was handled implicitly and occurred during export to the solver. Explicit RBE3 mesh creation at the time of meshing provides more information about the mechanics of your model, and allows you to edit and refine the mesh mating condition with greater confidence.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation**® connection mesh list® **Mesh Mating Condition**
Menu	**Insert**® **Model Preparation**® **Mesh Mating Condition**
Simulation Navigator	Right-click the FEM® **New Connection**® **Mesh Mating Condition**
Location in dialog box	**Parameters** group® **Mesh Mating Type** list

1D Connection

What is it?

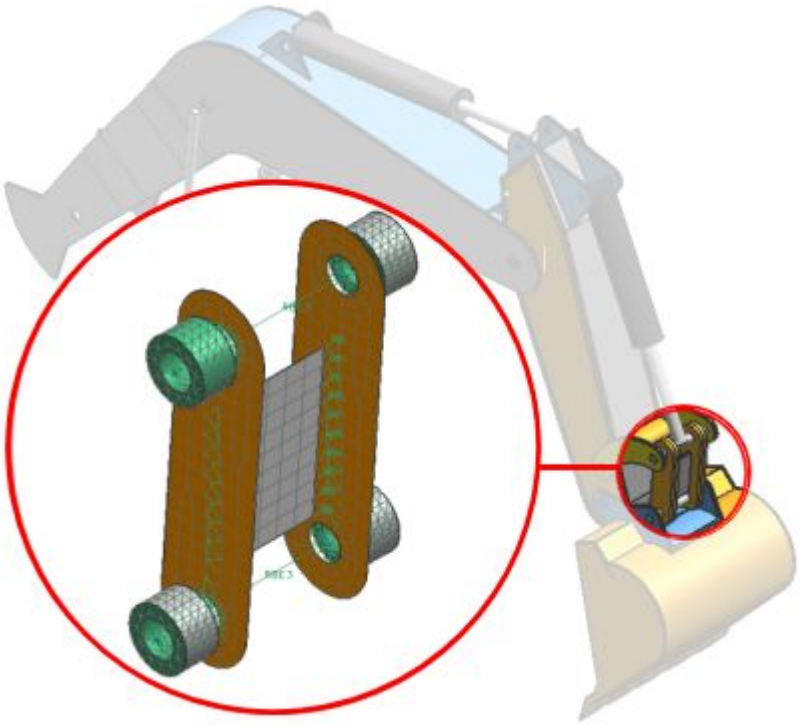
The **1D Connection** command replaces the following commands:

- **Edge-Face Connection**.
- **Create Spider Elements** (introduced in NX 5.0.1).
- The source–target methods in **1D Mesh**.

1D Connection is a more robust, flexible, and general-purpose tool for creating and managing connection meshes of 1D elements. The **1D Connection** command includes the following features:

- Support for connecting components in an assembly FEM.
- Support for both geometry-based and FE-based connections. Geometry-based connections are stored as mesh definitions until you mesh the connected bodies. Geometry-based connections are associative, and they update automatically when the connected meshes are modified.
- Expanded element type support.
- FE-based connection types: **Node to Node** and **Element Edge to Element Face**.
- Geometry-based connection types: **Point to Point**, **Point to Edge**, **Point to Face**, **Edge to Edge**, **Edge to Face**.

The following figure shows components in an assembly FEM joined by 1D connection elements. This model includes node-to-node, point-to-edge (spider elements), edge-to-face, and edge-to-edge connections.



Why should I use it?

Use **1D Connection** whenever you need to join meshes using 1D elements. You can use 1D connections to connect component FEMs within an assembly FEM, or to connect multiple sheet or solid body meshes within a FEM. Use 1D connections to define edge-face connections, spider and beam elements (to model pins or bolts), or to distribute mass.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation**® connection mesh list® **1D Connection**
Menu	**Insert**® **Mesh**® **1D Connection**

Spot Weld

What is it?

The **Spot Weld** command replaces the **Create Weld Elements** option found in the **1D Mesh** dialog box in previous releases. As in previous releases, you create spot welds by selecting points or edges to project, and specifying the top and bottom faces. Unlike the **Create Weld Element** dialog box, the **Spot Weld** dialog box enables you to specify mesh parameters and element properties at the same time.

See 1D Mesh process changes for more information.

Why should I use it?

Spot Weld provides a more consistent interface and more convenient workflow for creating 1D weld element meshes.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation**® connection mesh list® **Spot Weld**
Menu	**Insert**® **Mesh**® **Spot Weld**

Geometry abstraction

Stitching and unstitching edges

What is it?

Two new commands have been added to the **Model Cleanup** toolbar:

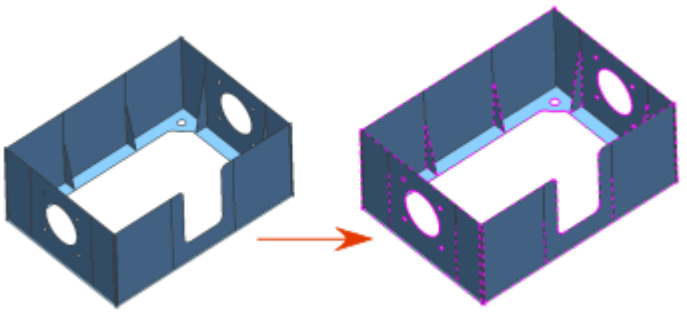
- **Stitch Edge**, which you can use to join two separate edges into a single edge or to stitch an edge into a face. **Stitch Edge** replaces the **Match Edge** command available in previous releases. **Match Edge** made two selected edges coincident, but they remained as separate edges.

- **Unstitch Edge**, which you can use to unstitch edges that you stitched together with the **Stitch Edge** command.

Stitch Edge is particularly useful for eliminating free edges that can occur when you create a midsurface on a thin-walled part.

Stitch Edge highlights free edges

When you select the **Stitch Edge** command, the software automatically highlights all free edges in your model. This makes it easy for you to identify edges that need to be stitched prior to meshing.



Different methods for stitching edges

The **Type** menu in the **Stitch Edge** dialog box controls how the edges are stitched.

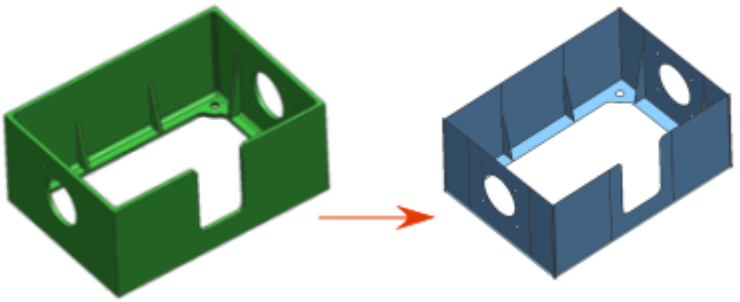
- Select **Automatic Free Edge to All Edges** to have the software automatically stitch all the free edges to other free edges within a specified search tolerance.

- Select **Manual Edge to Edge** to manually stitch a selected edge to another edge. This method is most similar to the **Match Edge** command from previous releases.

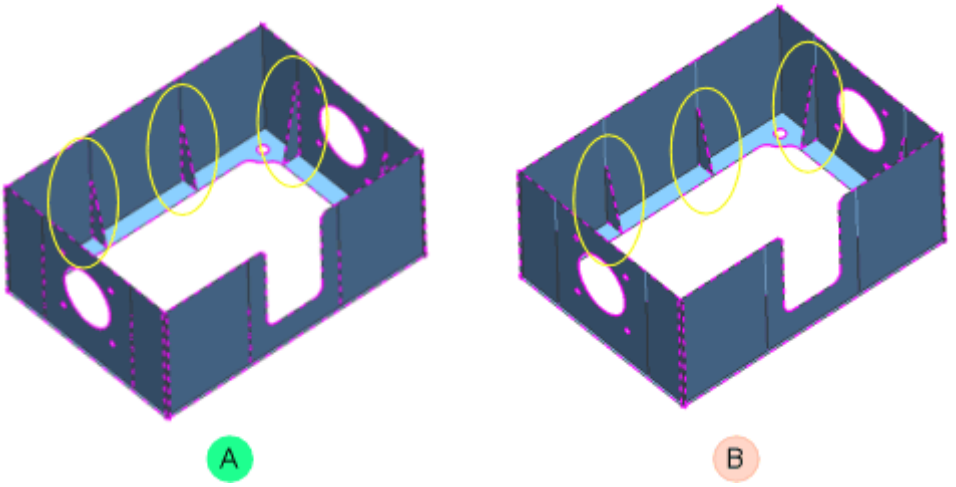
- Select **Manual Edge to Face** to manually stitch a selected edge into a face. This allows you, for example, to model T-type junctions as well as to connect ribs into the appropriate faces. This option replaces the **Match Meshes** option that was available in previous releases with the **Edge-Face Connection** command.

Stitch Edge examples

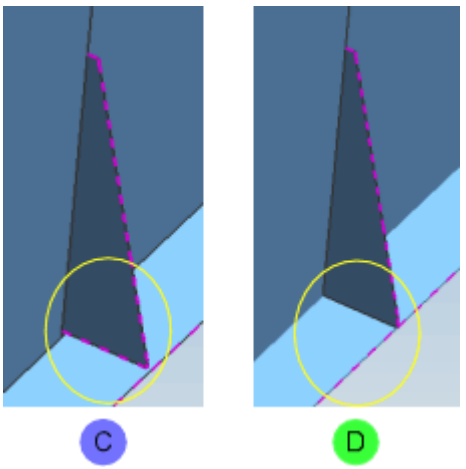
In the following example, we used the **Midsurface** command to generate a midsurface on a part.



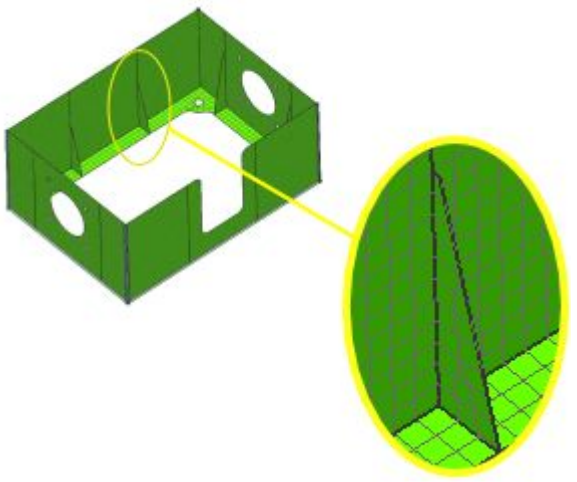
First, we used the **Stitch Edge** command with the **Automatic Free Edge to All Edges** option to stitch the free edges of the rib faces, shown in (A), into the wall faces. The result is shown in (B).



Next, we used the **Manual Edge to Face** option to manually stitch the bottom edges of the rib faces into the bottom face of the part. Notice how in (C), there is a gap between the bottom edge of the and the surface below. In (D), **Stitch Edge** eliminates this gap.



Finally, we generated a mesh on the part. Notice how the elements along the edges of the rib share common nodes with the elements on the surrounding faces.



Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Toolbar	**Model Cleanup**® **Stitch Edge** or **Unstitch Edge**
Menu	**Insert**® **Model Cleanup**® **Stitch Edge** or **Unstitch Edge**

New customer default for resetting model cleanup operations

What is it?

A new **Automatically Reset Geometry** option has been added to the General page of the **Meshing** customer defaults. Use this option to control how the software handles the deletion of any automatic or manual model cleanup operations (geometry abstractions) if you sufficiently reduce either the **Small Feature Tolerance** or **Element Size** for an existing mesh.

- If you select **Never**, the software preserves all abstractions. Any reduction you make in the **Small Feature Tolerance** or **Element Size** has no effect.
- If you select **Always**, the software deletes all existing abstractions, including any manual abstractions that you created with the commands on the **Model Cleanup** toolbar.
- If you select **Ask**, the software prompts you for confirmation before it deletes any abstractions.

Where do I find it?

Application	Advanced Simulation
Menu	**File**® **Utilities**® **Customer Defaults**®
Location in dialog box	**Simulation**® **Meshing**® General page

Boundary conditions

Multi-point constraints

What is it?

The **Manual Coupling** dialog box now includes an **MPC**, or multi-point constraint type, which lets you couple a single dependent node to one or more independent nodes. You define the degrees of freedom constrained for each of the dependent and independent nodes, as well as the coefficient of each node.

The **MPC** type replaces the **Constraint Equation** type, which could not be used as a true MPC as it had limited coupling behavior.

Note

When the solver is set to ABAQUS, the option name is ***EQUATION**. When the solver is set to ANSYS, the option name is **CE**.

For MSC Nastran 2007 and later, an additional type, **MPCY**, defines an MPC with a non-zero right-hand value for the MPC equation.

Why should I use it?

The new **MPC** option lets you create an MPC that is fully supported by the NX Nastran, MSC Nastran, ANSYS, and ABAQUS solvers.

You can now import MPCs from I-deas or other products into Advanced Simulation.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation**® **Manual Coupling**
Simulation Navigator	NX Nastran and MSC Nastran: Right-click the **Constraints** node under a Solution ® **New Constraint**® **Manual Coupling** Right-click the **Constraints Container** node® **New Constraint**® **Manual Coupling** ANSYS and ABAQUS: Right-click the **Simulation Objects** node under a Solution ® **New Simulation Object**® **Manual Coupling** Right-click the **Simulation Objects Container** node® **New Simulation Object**® **Manual Coupling**

Location in dialog box	NX Nastran and MSC Nastran: **Type** list ® **MPC** ABAQUS: **Type** list ® ***EQUATION** ANSYS: **Type** list ® **CE**

Support for Transient Initial Conditions

What is it?

You can now use the **Transient Initial Conditions** dialog box to define separate magnitudes for displacement and velocity at each degree of freedom, to be applied at the start of a structural transient analysis. You can use this constraint with NX Nastran and MSC Nastran solutions 109, 112, 129, 601 and 701 (it corresponds to the TIC bulk data entry).

For more information about the **Transient Initial Conditions** boundary condition, see Transient Initial Conditions overview in the Advanced Simulation help.

For more information about initial conditions in general, see the *NX Nastran Basic Nonlinear Analysis User's Guide*.

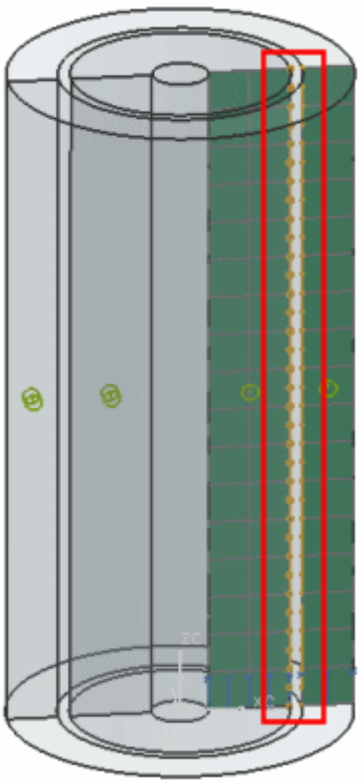
Where do I find it?

Application	Advanced Simulation
Prerequisite	Simulation file and the appropriate solution type active
Toolbar	**Advanced Simulation** toolbar® **Transient Initial Conditions**
Simulation Navigator	Right-click **Constraint Container**® **New Constraint**® **Transient Initial Conditions**

Advanced nonlinear contact for axisymmetric elements

What is it?

You can now use the **Advanced Axisymmetric Nonlinear Contact** dialog box to create a contact condition between axisymmetric elements in an NX Nastran advanced nonlinear solution. The axisymmetric 2D element types CQUADX4, CQUADX8, CTRAX3, and CTRAX6 are supported. This command is available for solutions 601,106 and 601,129.



The contact definition consists of a flexible source region that you can pair with either a flexible target region or a rigid target region. For example, you might use the rigid method when you have a steel die stamping a softer metal.

You define the source and target regions by selecting nodes along a face edge. A rigid target contact region must be a sequence of nodes not connected to elements.

For more information, see Advanced nonlinear contact for axisymmetric elements in the Advanced Simulation help.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Simulation file and the appropriate solution type active
Toolbar	**Advanced Simulation** toolbar® **Advanced Axisymmetric Nonlinear Contact**
Simulation Navigator	Right-click **Simulation Object Container**® **New Simulation Object**® **Advanced Axisymmetric Nonlinear Contact**

Validating the model

Checking node proximity to underlying geometry

What is it?

Use the new **Node Proximity to CAD Geometry** option in the **Model Check** dialog box to check the proximity of the nodes to the original CAD geometry. This lets you evaluate the fidelity of the mesh (which the software creates on the polygon geometry) to the underlying CAD part.

In the **Model Check** dialog box, you specify a **Proximity Tolerance** to define the maximum distance a node can lie from the corresponding CAD edge or face. When you click **OK** or **Apply**, the software evaluates the nodes and reports any that exceed the tolerance. If the software finds such nodes, you can use the **Adjust Node Proximity** option to move those nodes so they lie within the tolerance.

Note

You must have the idealized part loaded to use the **Node Proximity to CAD Geometry** option.

Why should I use it?

The **Node Proximity to CAD Geometry** check is most useful before you perform a contact analysis when you need to ensure the proximity of node locations in regions of contact. Prior to meshing the part, the software performs a number of automatic cleanup operations. These cleanup operations, for example, remove sliver surfaces and create seams in periodic faces. The software then generates the mesh. However, as a result of these cleanup operations, the nodes may not always be located exactly on the CAD surfaces. While small differences in node position are inconsequential for most types of analyses, they can be significant for the accurate detection of contact.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file that contains a mesh and the idealized part loaded
Toolbar	**Advanced Simulation® Finite Element Model Check**
Menu	**Analysis® Finite Element Model Check**

Checking solid properties of selected meshes or groups

What is it?

You can now use the **Solid Properties Check** command to verify the solid properties of a selected portion of your finite element model. In previous releases, you could only use **Solid Properties Check** to verify the solid properties of your entire model. Now, you can use **Solid Properties Check** to check the properties of either a selected mesh or group.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file
Simulation Navigator	Right-click a mesh or group® **Solid Properties**
Menu	**Information® Advanced Simulation® Solid Properties check**

Model Setup Check enhancements

What is it?

This release includes several enhancements to the **Model Setup Check** capabilities.

Solver version compatibility check for Nastran analyses

The **Model Setup Check** command now evaluates whether the entities you have defined in your model are compatible with the version of Nastran you specified on the Solve Options page of the **Solver Parameters** dialog box. Because the Advanced Simulation user interface is not specific to a particular solver version, it is possible to create, for example, loads or boundary conditions, that are not supported by the version of Nastran you are running. **Model Setup Check** issues a warning if it detects any entities in your model that are not consistent with the specified solver version.

Dependent degrees of freedom checks for Nastran models

With Nastran models, **Model Setup Check** now evaluates the validity of any dependent degrees of freedom for RBAR, RBE2, RBE3, and MPC bulk data entries:

- **Model Setup Check** verifies that a degree of freedom is never defined as a dependent degree of freedom (Nastran m-set) more than once. Such degrees of freedom are referred to as “double dependencies.”

Note

If you are using the NX Nastran solver, you can set the AUTOMPC parameter to Yes (using a **Solution Parameters** modeling object) to automatically resolve most double dependencies.

- **Model Setup Check** verifies that a dependent degree of freedom is never constrained (Nastran s-set) by an SPC, SPC1, or GRID bulk data entry.

Checks for assembly FEM labeling conflicts

With the addition of the new assembly FEM capabilities in this release, **Model Setup Check** now checks for node, element, and coordinate system labeling conflicts within a component FEM. If **Model Setup Check** detects conflicts, you can use the **Assembly Label Manager** to resolve them. See Assembly FEM for more information.

Checks for element associated data

Model Setup Check now evaluates the validity of any defined element associated data in your model. For example, if your model contains Nastran CQUAD4 elements with a specified material orientation vector, **Model Setup Check** verifies that the material orientation definitions are valid. See Ability to define properties on individual elements for more information, including a list of the supported associated data for each solver.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file
Toolbar	**Advanced Simulation**® **Model Check**
Simulation Navigator	Right-click a solution® **Model Setup Check**

Solvers and solutions

General solver enhancements

Supported solver versions

What is it?

In this release, Advanced Simulation supports the following solver versions:

- NX Nastran 6 and earlier versions
- MSC Nastran 2007 and earlier versions

- ABAQUS 6.7 and earlier versions
- ANSYS 11 and earlier versions

LS-DYNA support

What is it?

This release adds FEM support for the LS-DYNA solver. You can now create a FEM and then use **File® Export® Simulation** to write the model to an LS-DYNA keyword file.

In the LS-DYNA FEM, you describe the model's physical and material properties using the physical property named **PART** and the modeling objects named **SECTION** and **HOURGLASS**.

For example, in a physical property dialog box such as **PART**, you can specify an NX material or type the ID of an external LS-DYNA material.

You can define the material orientation for 2D and 3D elements in the physical properties, **Mesh Associated Data** dialog box, or the **Element Associated Data** dialog box.

The following table lists the LS-DYNA elements supported in Advanced Simulation. It also lists the keyword and physical property tables that can be defined for each element type. The SECTION modeling object contains additional physical properties such as shell thickness, spring constant, and so on. For a description of each element type, see the LS-DYNA documentation.

Element/LS-DYNA Keyword	Description	Mesh collector/Physical property name	Modeling object name
0D elements			
ELEMENT_MASS	0D structural mass element.	n/a	n/a
ELEMENT_INERTIA	Lumped inertia element assigned to a node.	n/a	n/a
1D elements			
ELEMENT_BEAM	Two-node 1D linear beam element (beam, truss).	PART (beam)	SECTION_BEAM HOURGLASS

ELEMENT_BEAM _OFFSET	Section properties (created automatically when you define offset and use default orientation).	PART (beam)	SECTION_BEAM HOURGLASS
ELEMENT_BEAM _ORIENTATION	Section properties (created automatically when you define orientation but not offset).	PART (beam)	SECTION_BEAM HOURGLASS
ELEMENT_BEAM _OFFSET _ORIENTATION	Section properties (created automatically when you define both orientation and offset).	PART (beam)	SECTION_BEAM HOURGLASS
ELEMENT_DISCRETE	Two-node 1D element (spring, damper).	PART (discrete)	SECTION_DISCRETE HOURGLASS
2D elements			
ELEMENT_SHELL (3), (4), (6), (8)	Three, four, six, and eight node 2D thin-shell elements.	PART (shell)	SECTION_SHELL HOURGLASS
ELEMENT_SHELL _THICKNESS	Thickness extracted from midsurface (created automatically).	PART (shell)	SECTION_SHELL HOURGLASS
ELEMENT_SHELL _OFFSET	Thickness offset (created automatically when you define offset).	PART (shell)	SECTION_SHELL HOURGLASS
ELEMENT_SHELL _BETA	Material orientation (created automatically when you define the angle in Element Associated Data).	PART (shell)	SECTION_SHELL HOURGLASS

ELEMENT_SHELL _MCID	Material orientation (created automatically when you define material coordinate system in Element Associated Data).	PART (shell)	SECTION_SHELL HOURGLASS
3D elements			
ELEMENT_TSHELL (6), (8)	Six-node and eight-node 3D solid Hex6 and Hex8 elements.	PART (thick shell)	SECTION_TSHELL HOURGLASS
ELEMENT_SOLID (4), (6), (8), (10)	Four, six, eight, and ten node 3D solid elements for isotropic materials.	PART (solid)	SECTION_SOLID HOURGLASS
ELEMENT_SOLID _ORTHO	Material orientation for orthotropic/anisotropic materials (created automaticaly when you define material orientation). Material orientation is defined by two vectors.	PART (solid)	SECTION_SOLID HOURGLASS

Note

LS-DYNA does not support solid wedge elements in parabolic form (15-node wedge). Therefore, you cannot extrude TRI6 parabolic shell elements in manual meshing.

LS-DYNA does not support Hex20 elements. Therefore, you cannot extrude QUAD8 elements in manual meshing.

Where do I find it?

Application	Advanced Simulation
Location in dialog box	Select **LSDYNA** as the **Solver** in the **New Fem** dialog box

Nastran support enhancements

NX Nastran Solution Monitor

What is it?

The NX Nastran **Solution Monitor** displays real-time information about the progress of the current solve. The information displayed depends on the version of NX Nastran you are running, your model, and the type of solution you are solving.

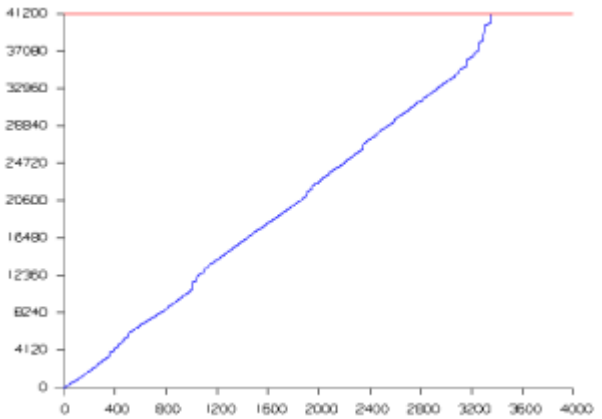
For all versions of NX Nastran, the **Solution Information** page on the **Solution Monitor** dialog box displays the contents of the .f04 file as they are written.

> **Note**
>
> Progress and convergence graphs are available only for NX Nastran 6.0 or later.

Progress and convergence provide additional information about the progress of your solution. The graphs displayed depend on the solution you are running.

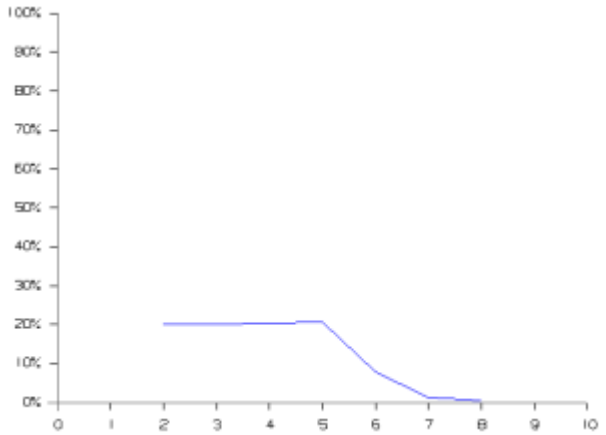
- The **Iterative Solver Convergence** monitor plots the convergence value against the iteration number.

- The **Sparse Matrix Solver** monitor plots the number of completed equations against the number of supernodes processed.



The sparse matrix monitor

- The **Contact Analysis Convergence** monitor plots the percentage of contact changes against the iteration number.



The contact analysis convergence monitor

- The **Eigenvalues Extraction** monitor graphs the number of eigenvalues extracted against the number of shift points. This monitor appears only when you use the Lanczos method to extract eigenvalues, and only when the software requires more than one shift point to extract the requested eigenvalues.

- The **DOF Curve** monitor plots the amplitude against time or frequency.

- The **Nonlinear History** monitor plots the load factor against the iteration number.

- The **Load Step Convergence** monitor plots the load factor against the iteration number.

Why should I use it?

As your solve progresses, use the **Solution Monitor** to monitor the convergence of the solution and estimate the remaining time required for the solution. Based on the monitor's output, you may want to cancel the solve, refine your model, or adjust solution parameters.

Where do I find it?

Application	Advanced Simulation
Prerequisite	You must run NX Nastran version 6.0 or later to view convergence graphs.
Toolbar	**Advanced Simulation**® **Solve**
Menu	**Analysis**® **Solve**
Simulation Navigator	Right-click a solution® **Solve**

Create and Edit Solution usability improvements

What is it?

This release includes improvements to the Nastran **Create Solution** and **Edit Solution** dialog boxes.

Create and Edit Solution dialog boxes reorganized

The options in the **Create Solution** and **Edit Solution** dialog boxes have been reorganized to make their correspondence to the different sections of the Nastran input file more clear. The options are now organized according to the section of the Nastran input file to which they belong. For example, the **Geometry Check** and **Max Job Time** options appear on the Executive Control page of options. This new organization more closely mirrors the structure of an actual Nastran input file and makes options easier to find.

Ability to preview a Nastran input file

Use the new **Preview Solution Setup** option in the **Create Solution** and **Edit Solution** dialog boxes to preview the associated Nastran solver input file in an information window. When you click **Preview**, the software displays the syntax of the input file based on the selected solution options. You can use the previewed file to validate that you have correctly specified all the necessary options for your analysis.

> **Note**
>
> The previewed input file is an abbreviated version of the actual input file. The software omits all node, element, load, and constraint data from the bulk data section.

Where do I find it?

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

Including comments or files in a Nastran input file

What is it?

You can now include specific text in a Nastran input file. Use the new **User Defined Text** type of modeling object to specify one of the following:

- The text to add and whether to add it to the beginning or end of the input file section.

- The name of the file to merge into the appropriate section of the input file.

If you want to include a file, the software inserts it in the appropriate section of the input file with a Nastran INCLUDE file statement. You use the **User Defined Text** option in the **Create Solution** or **Edit Solution** dialog box to specify the section of the input file in which to insert the **User Defined Text** modeling object.

Why should I use it?

With the ability to insert text or include entire files in your Nastran input file, you can, for example:

- Insert comments to help organize the data in your input file.
- Insert a file in the executive control section to include an alter to the solution sequence.

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**

Allocating DBset size

What is it?

Use the new **DBset Allocation** type of modeling object to create and control the maximum size of the following Nastran DBsets:

- DBALL (contains all DMAP data blocks that can be saved permanently for reuse in a subsequent analysis)
- SCRATCH (a temporary DBset for all scratch data blocks and files)
- SCR300 (the temporary workspace for the modules)

For all these DBsets, you can specify their maximum size in blocks, words, or bytes. For the SCRATCH and SCR300 DBsets, you can also specify a logical name and directory path.

If you include a **DBset Allocation** modeling object in your solution, when you solve your model, the software creates corresponding INIT and ASSIGN statements in the File Management section of your Nastran input file.

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**

Contact support enhancements

What is it?

Create a new **Contact Parameters** modeling object to define surface-to-surface contact parameters for Nastran SOL 101, 103, 111, and 112 analyses. When you solve your model, the software uses the **Contact Parameters** modeling object to create a Nastran BCTPARM bulk data entry in your NX Nastran input file. The contact control parameters on the **Contact Parameters** dialog box can help you adjust the contact algorithm when:

- You have trouble getting a solution to converge and complete.
- The contact results are not as you expected.

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**
Simulation Navigator	Right-click the Simulation® **New Solution**
Menu	**Insert® Modeling Objects**

Extended support for Nastran parameters

What is it?

This release includes improved support for Nastran parameters. You can now:

- Import parameter values specified on PARAM bulk data entries from either ASCII or op2 (binary) files.
- Use the new **User Defined** option in the **Solution Parameters** modeling object dialog box to specify other parameters that do not appear as options in the dialog box. This allows you, for example, to include custom parameters that you have created to work with a DMAP alter in your solution.

In Nastran, parameters are used extensively in the solution sequences to specify scalar values and request special features.

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**

Control over Nastran geometry check tolerance values

What is it?

Create a new **Geometry Check Options** modeling object to control the values that Nastran uses to evaluate the quality of the elements in your model prior to a solve. The options in the **Geometry Check Options** dialog box let you override Nastran's default element quality threshold values with reasonably modified values.

Note

You should always use good engineering judgment when you modify these default values.

Geometry Check Options control criteria for optional tests

The options that you specify in the **Geometry Check Options** dialog box control the number and severity of certain informational and warning messages produced by Nastran's optional geometry checks. These checks are different from a series of mandatory system-level checks that Nastran performs.

Nastran performs the system-level checks of the element quality regardless of whether you define any **Geometry Check Options**. There is no user control for the system controlled checks. The system controlled checks always produce a fatal error if Nastran finds an element that prevents the formulation of the finite element matrix.

Geometry Check Options are separate from NX element shape checks

The options in the **Geometry Check Options** dialog box are not related to the threshold values that the **Model Check** command uses to evaluate element quality. The element quality checks performed by the **Model Check** command evaluate different aspects of an element's deviation from an ideal size and shape. The **Model Check** command provides a broad assessment of element quality. However, different solvers require the elements meet different

quality threshold standards. Additionally, different solvers may use different techniques to evaluate an element's deviation from an ideal size and shape.

Because the **Model Check** command's criteria are not solver-specific, an element that passes the **Model Check** evaluations may fail a particular solver's element quality standards. With the Nastran solver, you can use a **Geometry Check Options** modeling object to loosen a particular tolerance value at solve time. However, the Nastran **Geometry Check Options** remain separate from the NX **Model Check** threshold values. Changes you make to the **Geometry Check Options** do not impact the **Model Check** threshold values, and vice versa.

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**
Simulation Navigator	Right-click the Simulation® **New Solution**
Menu	**Insert® Modeling Objects**

Support for modal effective mass output

What is it?

Use the options on the new Modal Effective Mass page in the **Structural Output Requests** dialog box to request the output of modal effective mass in normal modes analyses. These options let you request the output of modal effective mass, modal participation factors, and modal effective mass fractions. When you create a **Modal Effective Mass** type of output request, the software creates a MEFFMASS Case Control entry in the Case Control section of your Nastran input file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM or Simulation file
Toolbar	**Advanced Simulation® Modeling Objects**
Menu	**Insert® Modeling Objects**

Nonlinear analysis support

Support for additional nonlinear solution sequences

What is it?

This release includes support for an expanded range of Nastran nonlinear solution sequences.

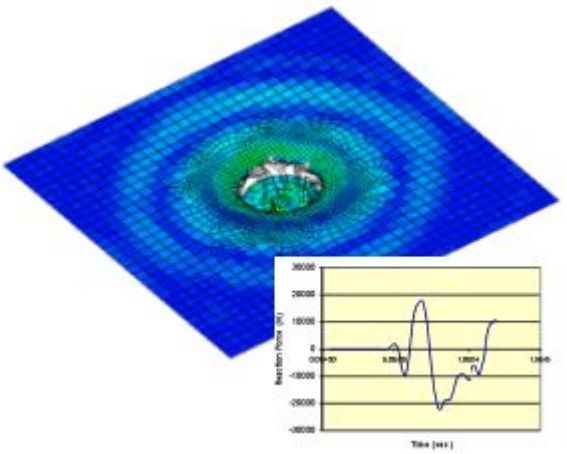
Support for SOL 129 analyses

With both NX Nastran and MSC Nastran, you can select the new **NLTRAN 129** option from the **Solution Menu** list in the **Create Solution** dialog box to create a SOL 129 (nonlinear transient response) solution. SOL 129 is primarily useful for dynamic transient response analysis, though it does also have limited static analysis capabilities. For more information on SOL 129 analyses, see the *NX Nastran Basic Nonlinear Analysis User's Guide*.

Support for SOL 701 analyses

With NX Nastran, you can select the new **ADVNL 701** option from the **Solution Menu** list in the **Create Solution** dialog box to create an Advanced Nonlinear SOL 701 solution. SOL 701 uses an explicit nonlinear solver, which is important for impact type problems that have a shorter time duration. SOL 701 is particularly useful for drop test simulations and metal forming analyses.

The following graphic shows the results of an impact analysis using SOL 701. Here, the impact against a plate creates a wave. The impact occurs in the first time step in the analysis, but the reaction force at the boundaries is not present until several steps later. The elements near the impact rupture and are removed from the analysis.



For more information on SOL 701 analyses, see the *NX Nastran Advanced Nonlinear Theory and Modeling Guide*.

Support for SOL 601 axisymmetric structural analyses

With NX Nastran, you can now perform advanced nonlinear type analyses on axisymmetric models. In the **Create Solution** dialog box, if you select **Axisymmetric Structural** from the **Analysis Type** list, you can now select **ADVNL 601,106** or **ADVNL 601,129** from the **Solution Type** list.

Axisymmetric analysis let you solve an FE model that is defined for only a section cut on one side of the axis of an axisymmetric part. This greatly reduces the degrees of freedom (DOF) and hence also significantly reduces solution time.

For more information on SOL 601 analyses, see the *NX Nastran Advanced Nonlinear Theory and Modeling Guide*.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with NX Nastran as the specified solver
Simulation Navigator	Right-click the Simulation® **New Solution**
Menu	**Insert**® **Solution**

Parameters for nonlinear analyses

What is it?

Three new types of solver-specific modeling objects have been added to allow you to define parameters for Nastran nonlinear analyses.

- Create a **Nonlinear Parameters** modeling object to define parameters that control the iteration strategy in the following types of Nastran analyses:
  - SOL 106, nonlinear static analysis (NLSTATIC 106)
  - SOL 153, steady-state, linear, or nonlinear thermal analysis (NLSCSH 153)

  With the NLPARM entry, the software uses a line search method to control the iterations. The options in the **Nonlinear Parameters** dialog box correspond to the fields in the NLPARM bulk data entry. For more information, see Nonlinear Parameters overview in the *Advanced Simulation* help.

- Create an **Arc-Length Methods Parameters** modeling object to control the arc-length incremental solution strategies in nonlinear static analysis (SOL 106/NLSTATIC 106). The options in the **Arc-Length Methods Parameters** dialog box correspond to the fields in the NLPCI bulk data entry.

To use the arc-length method, you must create an **Arc-Length Methods Parameters** modeling object in addition to the **Nonlinear Parameters** modeling object and associate them both with the solution. For more information, see Arc-Length Methods Parameters overview in the *Advanced Simulation* help.

- Create a **Nonlinear Transient Parameters** modeling object to define parameters that control the iteration strategy and time steps in nonlinear transient (SOL 129/NLTRANS 129) analyses. The options in the **Nonlinear Parameters** dialog box correspond to the fields in the TSTEPNL bulk data entry. For more information, see Nonlinear Transient Parameters overview in the *Advanced Simulation* help.

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** **Modeling Objects**
Simulation Navigator	Right-click the Simulation® **New Solution**
Menu	**Insert® Modeling Objects**

Strategy Parameters modeling object enhancements

What is it?

This release includes improvements to the **Strategy Parameters** modeling object.

- The **Strategy Parameters** dialog box now includes options that allow you to define time stepping for a SOL 701 (ADVNL 701) type analysis. These options are located on the new SOL Time Stepping page.

- The name of the corresponding field in the NXSTRAT bulk data entry now appears adjacent to each option in the **Strategy Parameters** dialog box. This helps clarify the correspondence between the numerous **Strategy Parameters** options and their analogous NXSTRAT fields.

- A new **Preview** button has been added to the **Strategy Parameters** dialog box. When you click **Preview**, the software displays the syntax of the NXSTRAT bulk data entry based on the currently selected options in an information window. You can use the previewed syntax to validate that you have correctly specified the appropriate options.

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**

Hyperelastic materials support

What is it?

Use the new **Hyperelastic-general** option in the **Type** list in the **Assign Material** dialog box to create hyperelastic materials for nonlinear analyses. These hyperelastic materials correspond to the Nastran MATHP bulk data entry. See Materials enhancements for more information.

Where do I find it?

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

Import/Export support

Import support for Nastran sets

What is it?

The SET case control command is now supported for import with both NX Nastran and MSC Nastran. Sets are now supported when you import the following case control output request commands:

ACCELERATION	FLUX	SPCFORCES
BCRESULTS	FORCE	STRAIN
DISPLACEMENT	GPFORCE	STRESS
EKE	OLOAD	THERMAL
ESE	MPCFORCES	VELOCITY

The software imports sets into either **Node Sets** or **Element Sets** in Advanced Simulation with the following limitations:

- You can only import a given SET ID once.
- The software imports the first occurrence of the SET command in the case control section.

Once you import the SET data into Advanced Simulation, you can use the node or element sets when you create new **Structural Output Request** and **Thermal Output Request** modeling objects. This allows you to generate results at specific nodes or for specific elements.

Sets categorized as node sets or element sets during import

To determine whether a set is a node or element set, the software looks at the output requests that reference the set.

- If the set is referenced by an output request that generates data at nodes, such as DISPLACEMENT, the software assumes the set is a node set.
- If the set is referenced by an output request that generates data on elements, such as STRESS, the software assumes the set is an element set.
- If the same set is referenced in the same subcase by different output requests that generate data at nodes and data on elements, the software assumes the set is a nodal set.

  Note

  Once the software determines that a set is either a node or element set, if that set is later referenced by the opposite type of output request (such as a node set referenced by an elemental output request, or vice versa), the software changes the associated **Entity** option from **Set** to **ALL**.

Where do I find it?

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

Import support for dynamic loads

What is it?

You can now import Nastran dynamic loads into Advanced Simulation. For example, this allows you to import frequency-dependent (RLOAD1) or time-dependent (TLOAD1) loads for use in a dynamic analysis.

- For a list of the specific case control commands and bulk data entries supported for import, see Nastran import support in the Advanced Simulation online help.
- For more information on dynamic loading in Nastran, see the *NX Nastran Basic Dynamics Analysis User's Guide*.

Where do I find it?

Application	Advanced Simulation

Menu **File® Import® Simulation**

Selective export for Nastran simulations

What is it?

For Nastran Simulation files, new options in the **Export Simulation** dialog box give you greater control over how the software exports your data.

- Use the new **Subset Export** option to control whether the software exports the entire model or just the visible meshes.
- Use the options in the new **Output Options** group to specify which sections of the Nastran input file to export. For the bulk data section of the input file, you can also control the specific entities exported. For example, if you only want to export node and element data, you can clear all check boxes except for **Grids** and **Elements**. This gives you flexibility over the amount of data you export.
- Use the options in the new **Entity ID Offsets** group to control the labels (IDs) for the entities you export. You can specify an offset for the labels of coordinate systems, nodes, elements, materials, and physical property tables.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file created with NX Nastran or MSC Nastran as the solver
Menu	**File® Export® Simulation**

Nastran import support enhancements

What is it?

This release includes a number of enhancements to the import support for Nastran bulk data entries and case control commands.

Newly supported bulk data entries and case control commands

Name	NX Nastran Import Support	MSC Nastran Import Support	Notes
BLSEG	Yes	No	The BY field is not currently supported.
CQUADX4	Yes	No	
CQUADX8	Yes	No	
CTRAX3	Yes	No	

CTRAX6	Yes	No	
DAREA	Yes	Yes	The DAREA entry is used for dynamic loading in SOLs 108, 109, 111, and 112. The software does not import this entry into a static solution.
EIGR	Yes	Yes	For the METHOD field, only the AHOU option is currently supported.
IC (case control command)	Yes	Yes	The MODAL and STATSUB fields are not currently supported.
MPC	Yes	Yes	See Multi-point constraints for more information.
MPCADD	Yes	Yes	See Multi-point constraints for more information.
MPCY	No	Yes	
NLPCI	Yes	Yes	
PLOADX1	Yes	Yes	• For the THETA field, only THETA=0.0 is supported. • PA must equal PB.
TABDMP1	Yes	Yes	
TABLED1	Yes	Yes	The XAXIS and YAXIS fields are not currently supported.
TABLED2	Yes	Yes	
TABLED3	Yes	Yes	
TIC	Yes	Yes	
TSTEPNL	Yes	Yes	

Enhancements for previously supported bulk data entries

Name	NX Nastran Import Support	MSC Nastran Import Support	Newly Supported Fields and Other Enhancements	Notes
BCTPARM	Yes	No	• PENTYP • You can now define NCHG as a real value	
BGPARM	Yes	No	PENN, PENT, PENTYP	
BOLT	Yes	No	You can now import this entry from both ASCII and op2 (binary) files.	An NX Nastran alter is currently required to write this information to the op2 file.

BOLTFOR	Yes	No	You can now import this entry from both ASCII and op2 (binary) files.	An NX Nastran alter is currently required to write this information to the op2 file.
CBAR	Yes	Yes	• PA and PB • The V vector is now supported as element associated data. See Ability to define properties on individual elements for more information.	
CBEAM	Yes	Yes	• PA and PB • The V vector is now supported as element associated data. See Ability to define properties on individual elements for more information.	
CBUSH	Yes	Yes	S, S1, S2, S3, and OCID fields	

CELAS2	Yes	Yes	• The K field now supported as element associated data. See Ability to define properties on individual elements for more information. • S	
CONM1	Yes	Yes	M41, M42, M43, M51, M52, M53, M61, M62, and M63	
CONM2	Yes	Yes	The M field is now supported as element associated data. See Ability to define properties on individual elements for more information.	
CQUAD4	Yes	Yes	THETA and MCID are now supported as element associated data. See Ability to define properties on individual elements for more information.	

CQUAD8	Yes	Yes	THETA and MCID are now supported as element associated data. See Ability to define properties on individual elements for more information.	If you try to import a CQUAD8 element that is missing any midside nodes, the software imports the element without any midside nodes at all
CQUADR	Yes	Yes	THETA and MCID are now supported as element associated data.	
CTETRA	Yes	Yes	Elements with missing midside nodes are now imported as specified in the solver input file.	
CTRIA3	Yes	Yes	THETA and MCID are now supported as element associated data. See Ability to define properties on individual elements for more information.	
CTRIA6	Yes	Yes	THETA and MCID are now supported as element associated data. See Ability to define properties on individual elements for more information.	If you try to import a CTRIA6 element that is missing any midside nodes, the software imports the element without any midside nodes at all

CTRIAR	Yes	Yes	THETA and MCID are now supported as element associated data.	
DLOAD	Yes	Yes	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.
FREQ	Yes	Yes	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.
FREQ1	Yes	Yes	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.
FREQ2	Yes	Yes	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.
FREQ3	Yes	Yes	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.
FREQ4	Yes	Yes	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.
FREQ5	Yes	Yes	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.
NLPARM	Yes	No	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.

RLOAD1	Yes	Yes	All fields supported.	You can now import this entry from both ASCII and op2 (binary) files.

Where do I find it?

Application	Advanced Simulation
Menu	**File® Import® Simulation**

Nastran export support enhancements

What is it?

This release includes a number of enhancements to the Nastran bulk data entries that are supported for export.

Newly supported bulk data entries

Name	NX Nastran (Design Simulation Version) Export Support	NX Nastran Export Support	MSC Nastran Export Support	Unsupported Fields or Options
BLSEG	No	Yes	No	All fields supported.
CQUADX4	No	Yes	No	All fields supported.
CQUADX8	No	Yes	No	All fields supported.
CTRAX3	No	Yes	No	All fields supported.
CTRAX6	No	Yes	No	All fields supported.
MPCADD	Yes	Yes	Yes	All fields supported.
MPCY	No	No	Yes	All fields supported.
NPLCI	No	Yes	Yes	All fields supported.
TIC	No	Yes	Yes	All fields supported. See Support for Transient Initial Conditions for more information.

Enhancements for previously supported bulk data entries

Name	NX Nastran (Design Simulation Version) Export Support	NX Nastran Export Support	MSC Nastran Export Support	Unsupported Fields or Options
CTRIAX6	No	Yes	Yes	All fields now supported.
MATHP	No	Yes	Yes	All fields now supported.
NLPARM	Limited	Yes	Yes	For the version of NX Nastran available in Design Simulation, all fields are unsupported except ID, NINC, and INTOUT.
TSTEP	No	Yes	Yes	All fields now supported.

Where do I find it?

Application	Advanced Simulation
Menu	**File® Export® Simulation**

ABAQUS support enhancements

ABAQUS keyword support enhancements

What is it?

The following table lists the new ABAQUS keywords that you can import and export along with the supported parameters for each keyword.

Keyword	Supported Parameters	Notes
*AMPLITUDE	NAME, TYPE=TABULAR	
*CONTACT PAIR	INTERACTION, ADJUST, EXTENSION ZONE, HCRIT, SMALL SLIDING, SMOOTH, TIED	
*ELSET	NAME	

*FRICTION	ELASTIC SLIP, LAGRANGE, ROUGH, SLIP TOLERANCE	
*NSET	NAME	
*SURFACE BEHAVIOR	AUGMENTED LAGRANGE, NO SEPARATION, PRESSURE-OVERCLOSURE	
*SURFACE INTERACTION	NAME	
*PRE-TENSION SECTION	NODE	In previous releases, this keyword was only supported for export.

Support for OP and AMPLITUDE parameters

This release also includes support for the OP and AMPLITUDE parameters for the *CLOAD, *DLOAD, *BOUNDARY, *TEMPERATURE, *CFLUX, *DFLUX, *CFILM, *DFILM, *SFILM, and *RADIATE keywords.

Support for new element types

This release also includes support for several new types of ABAQUS elements. See Newly supported element types for more information.

Where do I find it?

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

Selective export for ABAQUS simulations

What is it?

For ABAQUS Simulation files, new options in the **Export Simulation** dialog box give you greater control over how the software exports your data.

- Use the new **Subset Export** option to control whether the software exports the entire model or just the visible meshes.

- Use the options in the new **Output Options** group to specify which entities to include in the exported file. For example, if you only want to export node and element data, you can clear all check boxes except for **Nodes** and **Elements**. This gives you flexibility over the amount of data you export.

- Use the options in the new **Entity ID Offsets** group to control the labels (IDs) for the entities you export. You can specify an offset for the node, element, and material labels.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ABAQUS as the specified solver
Menu	**File® Export® Simulation**

New Contact Pair modeling object

What is it?

You can now create a **Contact Pair** modeling object to define properties for the pairs of surfaces that may contact or interact during the analysis. For example, the new **Contact Pair** modeling object lets you specify options to control:

- The default contact pressure-overclosure relationship.
- Friction properties.
- Surface interaction properties.

The options in the **Contact Pair** dialog box correspond to parameters for the ABAQUS *CONTACT PAIR, *SURFACE INTERACTION, *SURFACE BEHAVIOR, AND *FRICTION keywords.

Once you define a **Contact Pair** modeling object, you can reuse it with different **Surface-to-Surface Contact** simulation objects. In previous releases, you selected the **Contact Pair** option in the **Surface-to-Surface Contact** dialog box to define properties for the contacting surfaces. You had to define these properties each time you created a **Surface-to-Surface Contact** simulation object.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ABAQUS as the specified solver
Toolbar	**Advanced Simulation® Modeling Objects**
Menu	**Insert® Modeling Objects**

ANSYS support enhancements

New nonlinear analysis options

What is it?

When you are working with an ANSYS nonlinear statics analysis, new options on the Solution controls page of the **Create Solution** and **Edit Solution** dialog boxes give you greater control over how the software performs the analysis.

- Use the **SOLCONTROL** options to control the application of ANSYS' automatic solution methods. The optimized nonlinear default settings in ANSYS are designed to provide reliable and efficient settings for most analyses. However, if you are not satisfied with the results you obtain with the default settings, you can use the **SOLCONTROL** options to override them. See "SOLCONTROL" in the *ANSYS Commands Reference* manual for more information.

- Use the **NEQIT** options to limit the maximum number of equilibrium equations that the software performs at each substep. See "NEQIT" in the *ANSYS Commands Reference* manual for more information.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Menu	**Insert**® **Solution**
Simulation Navigator	Right-click the Simulation® **New Solution**, or right-click the current solution® **Edit Solution**

Cumulative options for controlling values of repeated loads and constraints

What is it?

A new Cumulative Options page has been added to the **Create Solution Step** and **Edit Solution Step** dialog boxes. These options allow you to use the ANSYS FCUM, SFCUM, BFCUM, and BFECUM commands to control whether repeated load or constraint values should replace previous values, be added to previous values, or be ignored in a given solution step.

For example, consider an analysis in which you apply a force to node 1 in the X direction as follows:

- In step 1, the value of the force is 200N.

- In step 2, the value of the force is 250N.

The **FCUM Command** controls the value of the force applied to node 1. From the **Forces/Moments in this step will** list:

- If you select **be added (ADD)**, the software adds the values of the forces from the previous steps and applies a force of 450N in step 2.

- If you select **replace previous value (REPLACE)**, the software applies a force of 250N in step 2.

- If you select **be ignored (IGNORE)**, the software ignores the force value defined for step 2 (250N) and instead applies the value of the force specified in the previous step, 200N.

If you select either **be added (ADD)** or **replace previous value (REPLACE)**, you can also specify a **Scale Factor** to the load or constraint value specified for the current load step. The software applies this **Scale Factor** to the current load value before it adds or replaces the load. For example, using the values from the previous example, if you select **be added (ADD)** and specify a **Scale Factor** value of 2.0, the software applies a load of 700N (2.0*250N +200N).

Note

Although ANSYS allows you to apply the FCUM, SFCUM, BFCUM, and BFECUM commands to specific node and element sets, Advanced Simulation applies the cFCUM, SFCUM, BFCUM, and BFECUM commands to all nodes and elements.

For more information on the FCUM, SFCUM, BFCUM, and BFECUM commands, see the *ANSYS Commands Reference* manual.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Menu	**Insert® Step**
Simulation Navigator	Right-click the active step and choose **Edit Solution Step**

New master degrees of freedom options for Householder algorithm

What is it?

In an ANSYS modal solution in which you are using the **Householder** algorithm, new options on the General page of the **Create Solution** and **Edit Solution** dialog boxes let you specify master degrees of freedom (DOFs). These master degrees of freedom control where the software reduces or condenses the mass matrix.

- Use the **Master Degrees of Freedom Definition (M)** options to manually select the master degrees of freedom and specify the active translational and rotational degrees of freedom. With this option, you specify the master degrees of freedom by selecting a **Node Set**. When you solve your model or export and ANSYS input file, the software creates an ANSYS M command.

  With the manual selection option, you can also use the **Total Number of Master DOF** box in the **Automatic Master Degrees of Freedom Generation (TOTAL)** options to specify the total number of master degrees of freedom to use in the analysis. This number should be greater than both the number of nodes in the selected **Node Set** and the specified **Number of Modes**.

- Use the **Automatic Master Degrees of Freedom Generation (TOTAL)** options to have the software automatically select the master degrees of freedom. With the automatic selection option, you simply specify the **Total Number of Master DOF**. This number should be at least twice the specified **Number of Modes**.

  You can also use the **Rotational Masters Key** option to specify whether to:

  – Include all degrees of freedom in the automatic selection.

  – Exclude rotational degrees of freedom from the automatic selection.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active Simulation file with ANSYS as the specified solver
Menu	**Insert**® **Solution**
Simulation Navigator	Right-click the Simulation® **New Solution**, or right-click the current solution® **Edit Solution**

Selective export for ANSYS simulations

What is it?

For ANSYS Simulation files, new options in the **Export Simulation** dialog box give you greater control over how the software exports your data.

- Use the new **Subset Export** option to control whether the software exports the entire model or just the visible meshes.

- Use the options in the new **Output Options** group to specify which entities to include in the exported file. For example, if you only want to export node and element data, you can clear all check boxes except for **Nodes** and **Elements**. This gives you flexibility over the amount of data you export.

- Use the options in the new **Entity ID Offsets** group to control the labels (IDs) for the entities you export. You can specify an offset for the labels of coordinate systems, nodes, elements, element types, materials, and real constants.

Where do I find it?

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

ANSYS command support enhancements

What is it?

This release includes improved import and export support for ANSYS commands and elements.

Import and export support for additional ANSYS commands

The following table lists the new ANSYS commands that you can import and export along with the supported parameters for each keyword.

Command	Supported Options or Arguments	Import Support	Export Support	Location in Advanced Simulation User Interface	For More Information
BFCUM	All options supported	No	Yes	Cumulative Options page in the **Create Solution Step** or **Edit Solution Step** dialog box	See Cumulative options for controlling values of repeated loads and constraints
BFECUM	All options supported	No	Yes	Cumulative Options page in the **Create Solution Step** or **Edit Solution Step** dialog box	See Cumulative options for controlling values of repeated loads and constraints

CE	All options supported	Yes	No	**Manual Coupling** dialog box, **CE** type option	See Multi-point constraints
CM	ENTITY=ELEM or NODE only	Yes	No	**Insert® Node and Element Set**	
CMBLOCK	ENTITY=ELEM or NODE only	Yes	No	**Insert® Node and Element Set**	
DCUM	All options supported	No	Yes	Cumulative Options page in the **Create Solution Step** or **Edit Solution Step** dialog box	See Cumulative options for controlling values of repeated loads and constraints
*DIM	TYPE=TABLE only	Yes	Yes	Table fields in loads and boundary condition commands	See Field enhancements
FCUM	All options supported	No	Yes	Cumulative Options page in the **Create Solution Step** or **Edit Solution Step** dialog box	See Cumulative options for controlling values of repeated loads and constraints
M	All options supported	No	Yes	Solution Control page in the **Create Solution Step** or **Edit Solution Step** dialog box for Structural,	See New master degrees of freedom options for Householder algorithm

				Modal analyses	
NEQIT	All options supported	No	Yes	Solution Control page in the **Create Solution Step** or **Edit Solution Step** dialog box for Nonlinear Static analyses	See New nonlinear analysis options
*SET	All arguments supported	Yes	Yes	Table fields in loads and boundary condition commands	See Field enhancements
SFCUM	All options supported	No	Yes	Cumulative Options page in the **Create Solution Step** or **Edit Solution Step** dialog box	See Cumulative options for controlling values of repeated loads and constraints
SOLCONTROL	All options supported	No	Yes	Solution Control page in the **Create Solution Step** or **Edit Solution Step** dialog box for Nonlinear Static analyses	See New nonlinear analysis options

TOTAL	All options supported	No	Yes	Solution Control page in the **Create Solution Step** or **Edit Solution Step** dialog box for Structural, Modal analyses	See New master degrees of freedom options for Householder algorithm

Import support for contact elements

Import support has been added for the TARGE170, CONTA173, and CONTA174 element types. The software imports these elements as a **Surface-to-Surface Contact** simulation object.

Support for new element types

This release also includes support for several new types of ANSYS elements. See Newly supported element types for more information.

Where do I find it?

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

NX Thermal and Flow, Electronic Systems Cooling, and Space Systems Thermal

General capabilities

Spatially varying boundary conditions

What is it?

In conjunction with the new fields functionality of Advanced Simulation, you can now define some loads, constraints, and simulation objects as varying in one, two, or three dimensions. For supported boundary condition types, you can define spatially varying values over any geometry supported for that type, except points. Since a point has no dimension, no spatially varying values can be defined on it.

For example, since you can define a heat flux on a polygon face or a polygon edge, you can define that flux as varying over the surface of the face, or the length of the edge.

You can define the variation by creating or linking to a table, or by creating a function to define the spatially varying fields. These fields can vary with time and space concurrently.

You can define the following boundary conditions as spatially varying:

- **Thermal Load**
- **Temperature**
- **Thermal Coupling**
- **Flow Boundary Condition**
- **Convection to Environment** (**Convection to Environment** type only)
- **Radiative Heating**
- **Advanced Thermal Coupling** (**One Way** type only)
- **Interface Resistance** (**Surface Interface** and **Edge Interface** types only)

  Note

  Available boundary conditions depend on solver licensing, and on the solver type and solution type you select.

Why should I use it?

In a physical model, known conditions typically vary spatially. Defining this spatial variation reflects the physical model more accurately than using a single value to represent the entire spatial domain for a boundary condition. This gives more accurate results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A supported boundary condition dialog box must be open
Location in dialog box	**Distribution** group® **Method** list® **Spatial**

Solution units

What is it?

You can now use options on the new Solution Units page in the **Create Solution** or **Edit Solution** dialog box to define the unit system for a solution. This sets the units that the solver uses in results files, message files, and report files. This does not affect boundary condition definition. Define the units by selecting one of these units systems from the **Solution Units** list:

- **Meter (newton)**
- **Foot (pound f)**
- **Meter (kilogram f)**
- **mm (milli newton)**
- **cm (centi newton)**
- **Inch (pound f)**
- **mm (kilogram f)**
- **mm (newton)**
- **Current part**

The units available on the **Temperature** list depend on which unit system you select for the solution. If you select an SI unit system (meter, mm, or cm), you can choose either **Celsius** or **Kelvin**. If you choose a non-SI unit system (foot or Inch), you can choose either **Fahrenheit** or **Rankine**.

When you select one of the unit systems, a list of the different physical units appears in the **Unit Information** box.

Why should I use it?

Having a choice of units when viewing results and solver files can make it easier to interpret them.

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click a solution ® **Edit Solution**
Location in dialog box	Solution Units page

Automatic naming of the run directory

What is it?

With the new Run Directory options in the Solution Details page of the **Create Solution** or **Edit Solution** dialog box, you can now allow NX to automatically name the run directory for a given solution when you run the analysis. You can specify the method to create the name by selecting an option from the **Run Directory Name** list:

- **Solution Name**: Names the directory to match the name of the active solution.

- **Current Simulation**: Uses the simulation directory. No run directory is created with this option.

- **Simulation – Solution Name**: Names the directory by combining the simulation name and the solution name, separated by a hyphen.

- **Specify**: Allows you to define the name and location of the run directory. (This is previously existing functionality.)

Why should I use it?

Allowing NX to automatically name the run directory ensures naming consistency and establishes a visible association between the analysis files and their solution and/or simulation.

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click a solution ® **Edit Solution**
Location in dialog box	Solution Details page® **Solve Options** group® **Run Directory** list

Initial Temperature and Initial 3D Flow Conditions replacement

What is it?

The new **Initial Conditions** constraint includes six types. Five of these types replace the **Initial 3D Flow Conditions** and **Initial Temperature** constraints that were available in previous releases.

The new **Initial Temperature** type replaces different previous functionality depending on the material properties of the elements it is applied to. If applied to:

- Thermal elements, it replaces the previous **Initial Temperature** constraint and is exactly equivalent.

- Fluid elements, it replaces the **Initial Temperature** option in the **Initial 3D Flow Conditions** constraint dialog box, and is exactly equivalent.

Four other types replace the correspondingly named options in the **Initial 3D Flow Conditions** constraint dialog box from previous releases, and are exactly equivalent. They are:

- **Initial Fluid Pressure**

- **Initial Fluid Velocity**
- **Initial Humidity** (available only for the **Advanced Flow**, **Advanced Thermal-Flow**, and **NX Advanced Thermal/Flow with ESC** solution types)
- **Initial Scalar** (available only for the **Advanced Flow**, **Advanced Thermal-Flow**, and **NX Advanced Thermal/Flow with ESC** solution types)

The sixth **Initial Conditions** type, **Initial Fluid Turbulence**, is completely new and had no counterpart in previous releases. For more details, see Initial Fluid Turbulence.

Where do I find it?

Application Toolbar	Advanced Simulation
Shortcut menu	**Advanced Simulation** toolbar ® **Initial Conditions** **Simulation Navigator**® right-click **Constraint Set** ® **New Constraint**® **Initial Conditions**
Location in dialog box	**Type** list

Variable Convection Coupling

What is it?

You can now model convection that varies with time or temperature as well as temperature difference using the **Convection Coupling** simulation object. Previously, only constant convection was supported. All types and convection parameter types are supported. You can also model convection that varies with mass flow rate or volume flow rate for the **Forced Convection Coupling** type only.

You set up the variation by defining a field with a unitless multiplier as the dependent variable and one of the supported model conditions as the independent variable. The software applies the multiplier to the convection parameter you define. You can also specify a multiplier that concurrently varies with time or temperature and with one of the other independent variables.

Why should I use it?

Thermal convection characteristics often change with time or varying thermal and flow conditions. Being able to define varying convection couplings broadens the range of applications for the **Convection Coupling** simulation object.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal or Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
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**® **Convection Coupling**
Toolbar	**Advanced Simulation** toolbar ® **Convection Coupling**
Location in dialog box	**Parameters** group® **Multiplier** list® **Field**® **Specify Field**

Extended element selection filtering

What is it?

New **Element Selection Filtering** options have been added to a number of boundary condition dialog boxes. When you are defining boundary conditions directly on elements, **Element Selection Filtering** makes it possible to select edges and faces of elements or to filter on a specific type of element.

> **Note**
>
> You must select **Element** from the main **Selection Filter** list to use the **Element Selection Filtering** options.

The available **Element Selection Filtering** options depend on the type of boundary condition you are creating. The options include:

- **0D Element**, **1D Element**, **2D Element**, and **3D Element**: Allow the selection of elements of the specified type only.
- **Element**: Allows selection of any type of element. You may select a combination of elements of different types.
- **Element Edges**: Allows the selection of 1D elements and element edges for 2D, and 3D elements.

- **Element Faces**: Allows the selection of 2D elements and faces of 3D elements.

Where do I find it?

Application	Advanced Simulation
Prerequisite	**Selection Filter** list set to **Element**.
Toolbar	**Advanced Simulation** toolbar ® Any supported boundary condition

Thermal Couplings that vary with temperature difference

What is it?

You can now define the **Magnitude** of the **Thermal Coupling** simulation object, or a **1 Way** type **Advanced Thermal Coupling** simulation object, based on varying temperature difference between the primary and secondary regions. In previous releases, the **Magnitude** of the coupling was variable with time or temperature.

To define a magnitude varying with temperature difference for the coupling, select the appropriate magnitude **Type** and then select **Field**. You can then select **Table Constructor** from the **Specify Field** list to define a table field with the independent variable as temperature difference.

For more information on enhancements to the fields capabilities, see Field enhancements.

Supported solvers and analysis types (Advanced Thermal Coupling)

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal or Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Note

The **Thermal Coupling** simulation object is available for all solution types.

Where do I find it?

Application	Advanced Simulation

Simulation Navigator	Right-click the simulation object container and select **New Simulation Object**® **Thermal Coupling** or **Advanced Thermal Coupling**
Toolbar	**Advanced Simulation** toolbar ® **Thermal Coupling** or **Advanced Thermal Coupling**
Location in dialog box	**Magnitude** group® **Field**® **Specify Field**

Time varying Interface Resistance simulation object

What is it?

Now you can specify a time varying **Magnitude** using **Fields** for the resistance of a **Surface Interface** or **Edge Interface** type **Interface Resistance** simulation object. In previous releases, you could define the **Magnitude** of the coupling only as a constant value or a mathematical expression.

- With the **Surface Interface** type, you can define the magnitude of the **Heat Transfer Coefficient**, **Total Conductance**, or **Total Resistance** as varying with time.

- With the **Edge Interface** type, you can define the magnitude of the **Total Conductance**, **Total Resistance**, or **Conductance per Length** as varying with time.

To define the magnitude as varying with time, select the appropriate magnitude **Type** and then select **Field**. You can then select **Table Constructor** from the **Specify Field** list to define a table field with the independent variable as time.

For more information on enhancements to the fields capabilities, see Field enhancements.

Why should I use it?

Being able to define time varying thermal resistance broadens the range of applications for the **Interface Resistance** simulation object. For example, you can create a time varying thermal resistance to simulate the thermal resistance of the bond between electronic components and the board.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the simulation object container ® **New Simulation Object**® **Interface Resistance**

Toolbar	**Advanced Simulation** toolbar ® **Interface Resistance**
Location in dialog box	**Magnitude** group® **Field**® **Specify Field**

Variable pressure rise for Duct Fan / Pump simulation object

What is it?

Now you can define the **Pressure Rise** mode for a **Duct Fan/Pump** simulation object to vary with volume flow rate or mass flow rate. If you select **Field** from the **Pressure Rise** list, you can select **Table Constructor** from the **Specify Field** list to define a table field to control the pressure rise variation.

For more information on enhancements to the fields capabilities, see Field enhancements.

Why should I use it?

The pressure rise for a Fan/Pump often changes with varying flow conditions. Being able to define varying pressure rise broadens the range of applications for the **Duct Fan/Pump** simulation object.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal or Advanced Axisymmetric Thermal
	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
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**® **Duct Flow Boundary Conditions**® **Duct Fan/Pump**
Toolbar	**Advanced Simulation** toolbar ® **Duct Flow Boundary Conditions** ® **Duct Fan/Pump**
Location in dialog box	**Parameters** group® **Mode** list® **Pressure Rise**® **Pressure Rise** list® **Field**® **Specify Field**

Interpolation scheme for nodal temperatures

What is it?

This release includes a new scheme for interpolating solver temperature output to nodes that is more accurate when you are viewing or interpreting nodal temperature results. It has no impact on viewing or interpreting element temperature results.

The interpolation is based on an adaptive piece-wise linear scheme that produces the greatest accuracy improvements in parts of the model where:

- Elements of different materials share nodes.
- Shell elements of different thicknesses share nodes.
- Shell elements in different planes share nodes.
- A thermal load or constraint is applied to elements surrounded by other elements rather than to elements at the boundary of the model.

This interpolation scheme provides a good balance between speed and reliability across a broad range of model configurations.

Generating additional reports after a solve

What is it?

Use the new **Refresh Reports** options on the Results Options page of the **Create Solution** or **Edit Solution** dialog box to generate additional reports without rerunning a solve. After you solve your model, you can create new **Report** simulation objects . You can then **Refresh Reports** to create the new reports from the existing solution data. This allows you to generate additional solution data while you are post-processing your results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An existing solution and new **Report** simulation objects
Simulation Navigator	Right-click a solution ® **Edit Solution**
Location in dialog box	Results Options page® **Control** group® **Refresh Reports**

Advanced radiation capabilities

Wavelength-dependent thermo-optical properties

What is it?

The **Thermo-Optical Properties — Advanced** type of modeling object has been enhanced to allow you to define non-gray thermo-optical properties of a surface. Non-gray thermo-optical properties vary as a function of wavelength.

Previously, all radiation/surface interaction was treated as either infrared or solar, depending solely on the radiative source, and assumed gray thermo-optical properties. Gray thermo-optical properties are invariable across the infrared or solar spectrum.

In NX 6.0, the same methods are valid. In addition, you can use a **Radiative Heating** simulation object to generate radiative energy of a specified wavelength or spectrum.

You can define these thermo-optical properties as a function of wavelength:

- **Non Gray Emissivity**
- **Specular Reflectivity**
- **Transparency**

Why should I use it?

In the physical world, the fraction of radiative energy absorbed, reflected and transmitted by surfaces depends on the spectrum, or wavelength distribution, of the radiation source. The spectral power variation with wavelength depends on the absolute temperature of the source.

In many applications, the temperature differences between the various surfaces, or more precisely the ratio of the absolute temperatures, are not large enough to justify the use of non-gray thermo-optical properties. In these cases, the gray surface approximation is valid.

For cases where the ratio of the absolute temperatures is significant, and optical properties vary significantly over wavelength, non-gray thermo-optical properties should be used. This often occurs in cryogenic applications or with high energy / high temperature heat sources like incandescent lamps.

Wavelength dependent thermo-optical properties are most useful when the simulation is performed using the spectral bands option. The wavelength-dependent properties are averaged over each radiation band in order to determine band-wise properties.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type

NX Thermal and Flow	Thermal	Advanced Thermal or Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Modeling Objects** ® **Thermo-Optical Properties—Advanced** ® **Create**
Menu	**Insert**® **Modeling Objects**® **Thermo-Optical Properties—Advanced** ® **Create**
Location in dialog box	**Properties** group® **Type** list ® **Non Gray—Wavelength Dependent** ® **Non Gray Properties** group

Wavelength-specific radiative heating

What is it?

You can define the spectral distribution of radiative energy emitted by a **Radiative Heating** simulation object. Previously, you could only specify the radiating energy as either infrared or solar spectrum. You define the distribution of energy as a function of wavelength by selecting **Specify Spectral Distribution** from the **Spectrum** list.

There are two methods to define spectral distribution of the energy, **Source Temperature** and **Intensity vs Wavelength**.

The **Source Temperature** method allows you to define wavelength by specifying the temperature of the emitting surfaces. The spectral distribution of the source will be that of a black body at the specified temperature. Following the black body approximation, high temperature objects emit radiative energy at a shorter wavelength than low temperature objects. By specifying a source temperature, you effectively specify the emission spectrum of the radiative source. Source temperature can vary with time.

The **Intensity vs Wavelength** method allows you to define a field with wavelength as the independent variable, and intensity as the dependent variable. Intensity is expressed as a fraction of the entire energy emitted. The specified intensity versus wavelength curve is normalized internally by the software over the spectrum, such that the total power specified for the source is respected.

Why should I use it?

In the physical world, radiating surfaces emit radiative energy in varying amounts at different wavelengths. For most simulation applications, this variation is not thermally significant, but when optical properties are strongly wavelength-dependent, substantial inaccuracies can result if the spectral distribution of sources are not taken into account when defining a radiative heat source. Two examples are cryogenic applications and halogen head lamps. Wavelength specific radiative heating is most useful when **Non-gray Thermo-Optical Properties** and multiple **Spectral Bands** are defined.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal or Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application Toolbar	Advanced Simulation
	Advanced Simulation toolbar ® **Radiative Heating**
Shortcut menu	**Simulation Navigator**® right-click the simulation object container® **New Simulation Object**® **Radiative Heating**
Location in dialog box	**Spectrum** group® **Spectrum** list

Spectral calculation bands

What is it?

Spectral Bands options have been added to the Radiation Parameters page of the **Solver Parameters** dialog box. These new options make it possible to improve accuracy and fidelity when calculating radiative heat exchange with wavelength dependent properties. The solver calculates radiation exchange in multiple discrete spectral bands, rather than using the two band approach. Use these new options to:

- Define spectral bands for calculation.
- Demarcate the transition between the infrared and solar spectra.

- Determine how bands are used in the infrared and solar regions of the spectrum.

Why should I use it?

Although default settings are often appropriate, expert users can define the wavelength discretization for improvements in accuracy or performance.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal or Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click a solution ® **Solver Parameters**
Location in dialog box	Radiation Parameters page ® **Spectral Bands** group

Segregated radiation solver

What is it?

The segregated radiation solver separates the solution of radiative heat transfer from the solution of all other thermal physics. The solver creates a coupled solution between a non-radiative conductance solver and the radiation solver. The results of the non-radiative conductance solver and the radiation solver are passed back and forth as boundary conditions until the solution converges.

The segregated radiation solver improves solver performance by:

- Reducing or eliminating radiative non-linearities.

- Segregating a sparse ill-conditioned solid conduction matrix from the dense radiation matrix. Ill-conditioning typically results from the solid matrix, which is much smaller than the radiative matrix. By segregating the two, the solver avoids having to compute high preconditioning fill values for a dense matrix that includes radiative conductances.

Note

The segregated radiation solver does not yet support models using the **Articulation** simulation object. In these cases, the default solver is used.

The software automatically uses the segregated radiation solver when a model includes the **Non Gray – Wavelength Dependent** type **Thermo-Optical Properties – Advanced** modelling object with multiple spectral bands. The segregated radiation solver can also be activated in the **Advanced Parameters** modeling object dialog box. For more information see Advanced Parameters overview.

Why should I use it?

The segregated radiation solver improves solver performance, particularly when the radiation matrix is very non-linear, or when the solid matrix is ill-conditioned. The improvement increases when you are modeling cryogenic temperatures.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Thermal or Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal or Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Thermal-Flow or Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling or NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation **Advanced Simulation** toolbar ® **Modeling Objects**
Toolbar	® **Advanced Parameters**® **Create**
Menu	**Insert**® **Modeling Objects**® **Advanced Parameters**® **Create**

Parallel view factor computations

What is it?

The new **Parallel Processing** options on the Solution Details page of the **Create Solution** or **Edit Solution** dialog box let you perform view factor computations in parallel on multiple computers or multiple processors. This can result in shorter solve times.

- If you select the **Hemicube Rendering** method for computing view factors from the **Calculation Method** list in the **Radiation** dialog box, you can specify multiple computers on which to run the analysis. Each computer must have a graphics card.

- If you select any other method for computing view factors, you can specify either multiple computers, or multiple processors on one or more computers.

If you select the **Run Solution in Parallel** option, you must then click **Validate Machines**. This launches a separate application that allows you to set up the environment for parallel processing. The software automatically balances the workload for computing view factors between the designated computers and/or processors.

Why should I use it?

For large radiation models, view factor calculations are often the most demanding process for a processor. Parallel processing can significantly reduce the solution time.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the active solution ® **Edit Solution**
Location in dialog box	Solution Details page® **Parallel Processing** group® **Run Solution in Parallel** check box

View factor performance improvement

What is it?

A new ray-tracing algorithm significantly improves the performance of a ray-traced view-factor calculation. Because view factor calculations are typically the most computationally intensive phase of analyses where radiation predominates, the new algorithm can significantly improve the solution time for large models where radiative heating or infrared exchange occurs.

The new ray-tracing algorithm uses an octree based spatial searching method. The octree ray-tracing algorithm is activated whenever ray tracing occurs. Ray tracing occurs either:

- In models whose surfaces have either specular or transparent optical properties.

- In models that contain at least one **Radiation**, **Solar Heating**, **Solar Heating Space**, **Orbital Heating**, or **Radiative Heating** simulation object in which **Monte Carlo** is the specified **Calculation Method**.

The octree ray-tracing algorithm is not yet supported in models that contain parabolic elements. Models with parabolic elements use the old ray-tracing method.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Thermal or Advanced Thermal
	Axisymmetric Thermal	Axisymmetric Thermal or Advanced Axisymmetric Thermal
	Coupled Thermal-Flow	Thermal-Flow or Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling or NX Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Prerequisite	A radiation model.

New radiation results types

What is it?

Now there are new results types in the **Radiation** group of the Result Options page of the Create Solution or Edit Solution dialog box. These new types output heat exchange data for infrared radiation between elements due to temperature difference. The result types produce radiation flux results that include both wavelength dependent and wavelength independent fluxes. The result types are:

- **Net Radiative Flux**: Computes scalar data equal to the net radiative flux into the element. The software calculates this value as the difference between the incoming and outgoing flux.
- **Radiosity Fluxes**: Computes scalar data equal to the sum of the emitted and reflected radiative fluxes out of the element.
- **Irradiance Fluxes**: Computes scalar data equal to the incident radiative flux on the element.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Prerequisite	A radiation model.
Simulation Navigator	Right-click a solution ® **Edit Solution**
Location in dialog box	Results Options page® **Radiation** group

Reorganized radiative heating flux results

What is it?

When you select certain radiative source flux result types, and your model includes multiple **Radiative Heating** simulation objects, the solver now outputs a separate data set for each **Radiative Heating** simulation object. The affected results types (**Radiative Source Fluxes** group, Results Options page of the **Create Solution** or **Edit Solution** dialog box) are:

- **Absorbed Fluxes**
- **Incident Fluxes**
- **Transmitted Fluxes**
- **Reflected Fluxes**

Previously, the solver generated a single data set for all **Radiative Heating** simulation objects.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Thermal	Advanced Thermal
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
NX Space Systems Thermal	Thermal	Space Systems Thermal

Where do I find it?

Application	Advanced Simulation
Prerequisite	A radiation model with multiple **Radiative Heating** simulation objects.
Simulation Navigator	Right-click a solution ® **Edit Solution**
Location in dialog box	Results Options page® **Radiative Source Fluxes** group or **Orbital and Radiative Source Fluxes** group (NX Space Systems Thermal)

3D flow capabilities

Periodic Boundary Condition enhancements

What is it?

Creating a **Periodic Boundary Condition** simulation object is now more intuitive. Instead of selecting a matching pair of nodes on the two periodic faces, you select the two periodic faces themselves. You designate one face as the master (independent) face and the other face as the slave (dependent) face.

For a **Translational** type periodic boundary condition, the **From Master to Slave** option on the new **Pressure Drop** list allows you to define a pressure drop between the master face and the slave face. The pressure drop is repeated at each subsequent periodic face.

Why should I use it?

The **Pressure drop between master and slave** option has applications for CFD analysis in industrial ovens and similar manufacturing installations.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click the simulation object container ® **New Simulation Object**® **Periodic Boundary Condition**
Toolbar	**Advanced Simulation** toolbar ® **Periodic Boundary Condition**

Fluid materials added to library

What is it?

The Advanced Simulation material library now includes defined **Air** and **Water** materials with properties that vary with temperature. In previous versions of the material library, **Air** and **Water** were defined with constant properties.

The material library includes a total of 34 new fluid materials. All the new fluid materials are defined with temperature dependent **Mass Density**, **Thermal Conductivity**, **Dynamic Viscosity**, and **Specific Heat Pressure**. If appropriate, they are also defined with a temperature dependent **Thermal Expansion Coefficient**.

The new fluid materials are:

103 Ammonia_NH3Liquid
104 Carbon_Dioxide_Liquid
105 Sulfur_Dioxide_Liquid
106 Glycerin_Liquid
107 Freon_Liquid_R12
108 Ethylene_Glycol_Liquid
109 Engine_Oil_Liquid
110 Mercury_Liquid
111 Helium_Gas
112 Hydrogen_Gas_H2
113 Oxygen_Gas_O2
114 Carbon_Dioxide_Gas
115 Ammonia_Gas
116 Water_vapour_Gas
117 Air_Temp-dependent_Gas
118 Water_saturated_Liquid
119 Nitrogen_Gas_N2
120 Bismuth_Liquid
121 Lead_Liquid
122 Potassium_Liquid
123 Sodium_Liquid
124 NaK(45-55)_Liquid
125 Nak(22-78)_Liquid
126 PbBi(45-55)_Liquid
127 Acetylene_C2H2_Gas
128 Argon_Ar_Gas
129 Methane_CH4_Gas
130 Propane_C3H8_Gas
131 Methanol_CH3(OH)
132 Isobutane_(R600a)_Liq
133 R134a_C2H2F4_Liquid
134 R134a_C2H2F4_Gas
135 Acetylene_C2H2_Liquid
136 Isobutane_(R600a)_Gas

Where do I find it?

Application	Advanced Simulation
Prerequisite	FEM, idealized part, or part active
Toolbar	**Advanced Simulation® Material Properties**
Location in dialog box	**Choose Material**

Ignore specific obstructions when meshing a fluid domain

What is it?

When you use the Fluid Domain simulation object command to generate a fluid mesh, you can now specify that a selected solid or sheet body within the volume of a fluid domain mesh should be ignored by the software. Since they are within the volume of the fluid domain mesh, these bodies would otherwise be treated as obstructions to the fluid mesh. With the **Inner Regions to Mesh Through** group active, select any body that should not be treated as an obstruction to the fluid mesh. The software then meshes through this body rather than around it.

Why should I use it?

Previously, when a body within the volume of a fluid domain did not model a physical obstruction to the flow, you had to either remove it, or separately mesh it with fluid elements. The **Inner Regions to Mesh Through** option allows you to easily create an efficient mesh through these bodies.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Flow or Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow or Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling or NX Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click the simulation object container® **New Simulation Object**® **Fluid Domain**
Toolbar	**Advanced Simulation** toolbar ® **Fluid Domain**

K-omega turbulence model

What is it?

You now have a sixth option, **K-omega**, when selecting a turbulence (viscous) model on the Solution Details page of the **Create Solution** or **Edit Solution** dialog box. The K-omega turbulence model is a Reynold's averaged Navier Stokes turbulence model. It is a two equation model which models both the transport and production of, and the specific dissipation of turbulent kinetic energy. It is similar in form to the K-epsilon model, although the terms of

the K-omega model are often argued to have a stronger fundamental physical basis. This model is similar in many respects to the Shear Stress Transport (SST) model, which essentially uses the K-omega model near the wall and the K-epsilon model away from the wall.

Why should I use it?

The K-omega model has been widely adopted by both industry and academia. Consider using the K-omega model in situations where you want to have a good turbulence model which predicts effects for both wall-bounded and free shear flows. Like the SST model, the K-omega model can be used without wall functions, allowing you to integrate the flow solution to the wall if you have a very detailed skin mesh. This can yield a very accurate resolution of the boundary layer.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click a solution ® **Edit Solution**
Location in dialog box	Solution Details page® **Solve Options** group® **Turbulence Model** list

Deactivate or activate the wall function

What is it?

The **Use Wall Function** check box on the 3D Flow page of the **Create Solution** or **Edit Solution** dialog box lets you:

- Globally deactivate the wall function for friction calculations.
- Apply the turbulence equations all the way to the wall for every surface where friction is calculated.

The option is only applicable when you also select **K-omega** or **Shear Stress Transport (SST)** from the **Turbulence Model** list on the Solution Details page of the **Create Solution** or **Edit Solution** dialog boxes. For all other turbulence models, the option has no effect.

The **Use Wall Function** check box in the **Flow Surface** and **Flow Blockage** dialog boxes lets you control the application of the wall function locally. The **Use Wall Function** option in the **Flow Surface** or **Flow Blockage** dialog box overrides the global setting for the selected geometry.

Why should I use it?

Clear the **Use Wall Function** check box when you use the K-omega or SST turbulence model if you need a very accurate resolution of the boundary layer. This approach requires a finer mesh near the wall, preferably a detailed skin mesh.

If you need a faster solution time, or if a detailed analysis of the boundary layer is unimportant, select the **Use Wall Function** check box.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Flow, Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow, Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling, Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click a solution ® **Edit Solution**
Location in dialog box	Solution Details page® **Solve Options** group® **Turbulence Model** list

Initial turbulence

What is it?

You can specify initial turbulence in your model globally in the Initial Conditions page of the **Create Solution** or **Edit Solution** dialog box, or locally in the new **Initial Conditions** constraint dialog box.

The **Initial Turbulence** type of **Initial Conditions** constraint defines the turbulent energy of the entire fluid domain, or in a selected fluid region in one of three ways:

- **Intensity and Length Scale** (Specify **Turbulent Intensity** and **Eddy Length**.)
- **K-epsilon** (Specify **Turbulent Kinetic Energy** and **Dissipation Rate**.)

- **K-omega** (Specify **Turbulent Kinetic Energy** and **Specific Dissipation Rate**.)

The three methods are provided for convenience only, since they are equivalent. Use whichever method matches the data you want to use.

Why should I use it?

For a transient model, specifying initial turbulence in your model, or in a specific region in your model, can give more accurate results in the first few time steps.

For a steady state model, specifying an approximate initial turbulence in your model, or in a specific region in your model, can accelerate convergence.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Flow or Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow or Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling or NX Advanced Thermal/Flow with ESC

Where do I find it?

Global initial turbulence

Application	Advanced Simulation
Prerequisite	In the **Initial Conditions** page of the **Create Solution** or **Edit Solution** dialog box, **Initial Conditions** list® **Uniform**
Shortcut menu	**Simulation Navigator**® right-click a solution® **Edit Solution**
Location in dialog box	Initial Conditions page® **Turbulence Characteristics** list

Local initial turbulence

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click the constraint set container® **New Constraint**® **Initial Conditions**
Toolbar	**Advanced Simulation** toolbar ® **Initial Conditions**
Location in dialog box	**Type** list

Porous Flow Blockage enhancements

What is it?

This release includes several enhancements to the **Flow Blockage** simulation object.

New option for defining whether flow is laminar through the blockage

If you select either **Porous Blockage-Isotropic** or **Porous Blockage-Orthotropic** from the **Type** list, you can use the new **Flow is Laminar in Blockage** option to specify whether the flow through the blockage is laminar.

- Select **Flow is Laminar in Blockage** to indicate that the flow through the blockage is laminar.
- Clear the **Flow is Laminar in Blockage** check box to indicate that the flow through the blockage is the same as the flow in the fluid domain.

New methods for calculating flow resistance for isotropic porous blockages

If you select **Porous Blockage-Isotropic** from the **Type** list, several new methods for calculating flow resistance have been added to the **Specify Method** list:

- **Pressure Drop per Length** lets you specify the pressure drop through an isotropic porous blockage as a function of velocity.
- **Packed Bed of Spheres** lets you simulate flow through a 3D filter, such as HEPA type filters.
- **Fibrous Porous Media** lets you simulate flow through packed bed reactors.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Flow or Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow or Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling or NX Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click the simulation object container® **New Simulation Object**® **Flow Blockage**

Toolbar	**Advanced Simulation** toolbar ® **Flow Blockage**
Location in dialog box	**Type** list

Particle tracking analysis

What is it?

The flow solver can now track the movement of particles through the fluid domain. You can define the location at which the particles are injected into the fluid and their physical characteristics by creating a **Particle Injection** simulation object on the inflow boundary of your fluid domain.

Particle tracking analysis can be simulated with the transient or steady state flow simulation. For the steady state simulation, you can define parameters for the calculation of the nominal output frequency in the **Steady State Particle Controls** group of the 3D Flow Solver page on the **Solver Parameters** dialog box.

Why should I use it?

You should use particle tracking when you need to simulate how particles are dispersed in a fluid domain.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Shortcut menu	**Simulation Navigator**® right-click the simulation object container® **New Simulation Object**® **Particle Injection**
Toolbar	**Advanced Simulation** toolbar ® **Particle Injection**

Acoustic power density as a flow result data set

What is it?

You can select the new **Acoustic Power Density** option on the Results Options page of the **Create Solution** or **Edit Solution** dialog box to quantify the acoustic noise generated by the simulated turbulent flow. **Acoustic Power Density** estimates the acoustic power of the isotropic turbulent motion of a fluid. The data generated is based on Lighthill's acoustic theory and uses an analytical correlation developed by Proudman.

> **Note**
>
> When you select **Acoustic Power Density** you must select **K-Epsilon** , **K-Omega** or **Shear Stress Transport – SST** from the Turbulence Model list (Solution Details page of the **Create Solution** or **Edit Solution** dialog box) to generate the acoustic power data. If you do not select one of these turbulence models, the software issues a warning and does not generate the **Acoustic Power Density** result.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC

Where do I find it?

Application	Advanced Simulation
Prerequisite	A flow solution with the appropriate **Turbulence Model** specified
Simulation Navigator	Right-click a solution ® **Edit Solution**
Location in dialog box	Results Options page® **3D Flow** group® **Acoustic Power Density**

Carreau non-Newtonian model

What is it?

When you use a **Power-Law** type of **Non-Newtonian Fluid** modeling object, you can now use the Carreau model to describe pseudoplastic non-Newtonian fluids, such as ketchup, whipped cream, blood, paint, nail polish, polymer solutions, and molten polymers.

You use an **Advanced Parameters** modeling object to specify the Carreau model. For complete instructions on how to use the Carreau non-Newtonian

model, see Define a non-Newtonian pseudoplastic fluid using the Carreau model.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Advanced Flow
	Coupled Thermal-Flow	Advanced Thermal-Flow
NX Electronic Systems Cooling	Coupled Thermal-Flow	NX Advanced Thermal/Flow with ESC

Where do I find it?

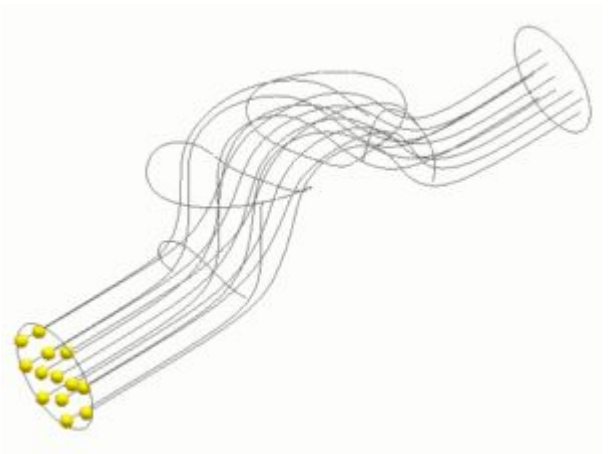
Application	Advanced Simulation **Advanced Simulation** toolbar ® **Modeling Objects**
Toolbar	® **Advanced Parameters**® **Create**
Menu	**Insert**® **Modeling Objects**® **Advanced Parameters**® **Create**

Post-processing

Streamlines

What is it?

Streamlines are a post-processing display type that represents velocity results by showing the path taken by a massless particle from a defined point. Each point along the streamline is tangent to the velocity vector of the fluid flow.



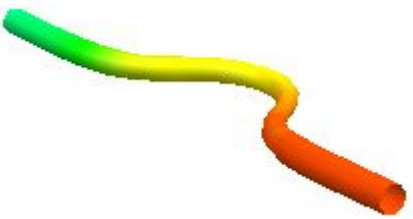
To create a streamline display, you must define one or more seed points. Seed points are stored in seed sets. Seed sets are similar to the paths used for graphing results; you can reuse seed sets in multiple displays, and manage seed sets in the **Post-Processing Navigator**.

There are four styles of streamline:

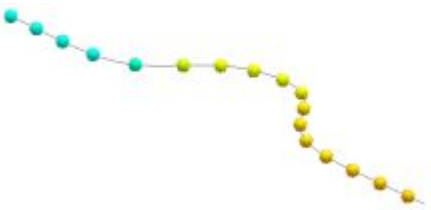
- **Line** — A simple streamline display. You can plot selected results values on streamlines as smooth contours.

- **Ribbon** — Displays each streamline as a constant-width ribbon, where the twist of the ribbon represents the vorticity of the flow.

- **Tube** — Displays each streamline as a tube, where the radius of the tube indicates the divergence of the flow.



- **Bubbles** — Displays spheres along the streamline, at a specified time increment. Bubbles are also used for animating steady-state streamlines.



Why should I use it?

Typically, streamlines are used to present results of a flow analysis. Use streamlines to improve the interpretation and communication of velocity results.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A solved model with velocity results A displayed post view
Toolbar	**Post Processing** toolbar® **Post View**
Menu	**Tools**® **Results**® **Post View**
Post-Processing Navigator	Right-click a post view® **Edit** Right-click a post view® **Create Streamlines** **Display** tab® **Streamlines** (display type list)® **Result**
Location in dialog box	**Display** tab® **Streamlines** (display type list)® **Streamline Parameters**® **Options**

Groups support

What is it?

Post-processing now includes support for groups:

- You can display results on selected groups.

  In the **Post-Processing Navigator**, expand the post view node, expand the **Groups** node, and select the group to display. You can control the scaling of the color bar on groups using the **Results** and **Displayed** options on the Color Bar page of the **Post View** dialog box.

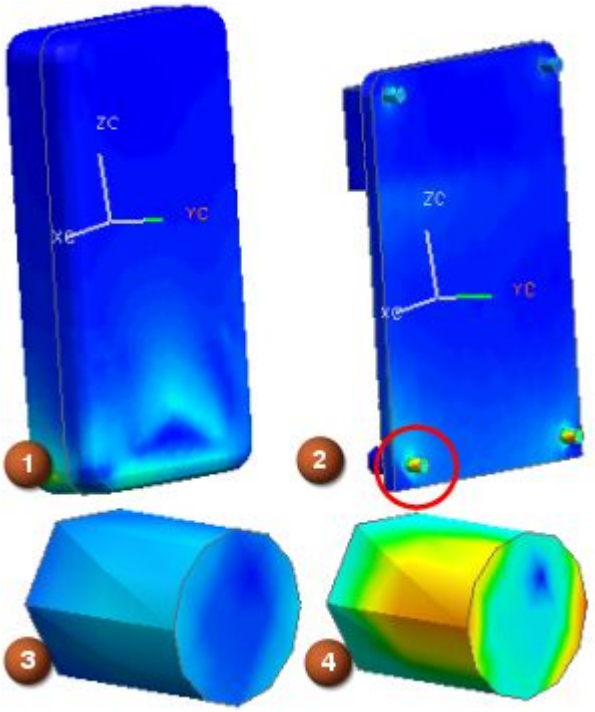
- You can identify results on a specified group.

  In the **Identify** dialog box, choose **Nodes in Group** or **Elements in Group** (depending on the result type) from the results list. The software lists all element, nodal, or element-nodal results in the specified group.

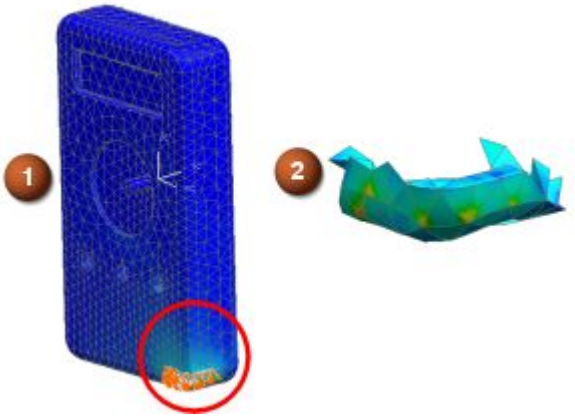
- You can save identified results to a group.

  For example, you can identify the 10 highest element-nodal results values, and save the identified elements to a group. This group is saved with the Simulation file. In the **Identify** dialog box, click **Save Selection in Group** .

For more information, see FE Groups.



Four post views showing (1) the complete model; (2) an interior mesh, with a stored element group representing a boss (red circle); (3) results displayed on the element group, with Color Bar set to Results; and (4) results displayed on the element group, with Color Bar set to Displayed.



(1) The 50 highest results are identified (highlighted in the red circle) and saved to a group. (2) Results are displayed on the element group. Note that the resulting group includes elements from multiple meshes.

Why should I use it?

You can create groups representing areas of interest in your model and view results on just those elements. By setting the color bar scale to **Displayed**, you can visualize results on these areas of interest with greater resolution.

You can also use groups to quickly identify results in areas of interest. And by saving identified results to a group, you can quickly isolate these areas for further refinement, or compare results from different solutions.

Where do I find it?

Displaying results on groups

Application	Advanced Simulation **Post-Processing Navigator**® **Fringe Plots**® post view node® **Groups** node® right-click a group® **Show Only**
	Post-Processing Navigator® **Fringe Plots**® post view node® **Groups** node® right-click a group® **Show**
Shortcut menu	**Post-Processing Navigator**® **Fringe Plots**® post view node® **Groups** node® right-click a group® **Hide**

Identify

Application Toolbar	Advanced Simulation **Post Processing** toolbar® **Identify**
Shortcut menu	**Post-Processing Navigator**® **Fringe Plots**® right-click a post view® **Identify** **Identify** dialog box® results list® **Elements in Group** **Identify** dialog box® results list® **Nodes in Group**
Location in dialog box	**Identify** dialog box® **Save Selection in Group**

Composite Laminates

Composite Laminates toolbar

What is it?

The **Laminates** toolbar contains frequently used laminates commands. The available commands depend on whether you are working in a FEM or Simulation file.

In a FEM file, the following commands are available:

Command	Description
Physical Properties	Creates a laminate physical property.
Global Layup	Creates a global layup for laminate ply based modeling. See Ply-based laminate modeling for more information.
Ply Materials	Creates and manages ply materials.
Update Global Layups and Zones	Drapes plies, updates fiber orientations, and recomputes zones.

In a Simulation, the following commands are available:

Command	Description
Spreadsheet Post Report	Generates a spreadsheet report of laminate results.
Graphical Post Report	Generates a graphical report of laminate results. See Graphical post processing for more information.

Where do I find it?

Application Advanced Simulation

Laminates built from plies

Ply-based laminate modeling

What is it?

You can now define laminate properties with the alternate **Global Layup** method. The new **Layup Modeler** dialog box, lets you define global ply stacking sequences. The global plies can be assigned directly to CAD surfaces and appear as objects in the **Simulation Navigator**. You can create as many layups and global plies as you like, but a global ply has a unique ID and can only exist in a single layup.

Draping algorithms compute ply fiber distortions on undevelopable surfaces, and fiber orientations on all types of surfaces. When you define ply draping data, the software creates sets of elements called **Zones** that share similar ply orientations. These zones point to automatically created laminate physical properties. For more information, see Surface Selection, draping and zones.

You enable the **Global Layup** approach at the mesh collector level by selecting the **Inherit from Layup** stacking recipe in the collector's laminate physical property.

Why should I use it?

In ply-based layup modeling, you assign plies to regions of the model, rather than assigning an entire laminate layup to a mesh collector. This provides a modeling flexibility more appropriate to some industrial applications, such as using hand layups.

Ply-based layup modeling more closely resembles the process of hand layups. For example, with ply-based layup modeling you can add plies to certain regions and omit plies from other regions.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with a 2D mesh on the geometry
Toolbar	**Laminate** toolbar® **Global Layup**
Menu	**Insert**® **Laminate**® **Global Layup**

Surface selection, draping, and zones

What is it?

You can assign a selected ply or plies to a selected face or faces and define draping parameters in the new **Draping Data** dialog box. Draping simulates how fiber orientation is affected when you "drape" an orthotropic ply over a non-planar surface, or a set of surfaces not in the same plane.

Depending on the model's geometry, the calculated ply directions on adjoining draped faces can affect each other. In the **Ply draping data** dialog box, select an edge as a **Cut Curve** to separate two adjoining faces so that their ply directions are calculated separately.

The software calculates ply direction and distortion due to draping when you right-click the layup node in the **Simulation Navigator** and select **Update**.

The draped ply orientations and any defined material orientations are used to compute **Zones**, which are sets of 2D elements sharing a unique laminate physical property. Laminate ply angle is determined by draping data and expressed as a rotation from the element material orientation.

While a typical zone contains many elements, each zone can be as small as a single element. Zones are smaller when there is:

- Strong curvature.
- Small zone angle tolerance.
- A different material orientation for each element.

The software creates zones automatically, and they can be ignored if draping is simple or the draping results are well understood. They are most useful for verifying correct layup and material orientation, or when draping is complex.

Where do I find it?

Define surface selection, draping, and zones from the **Simulation Navigator**:

Application	Advanced Simulation
Prerequisite	A ply defined and selected
Simulation Navigator	Right-click the ply node® **Edit**

Define surface selection, draping, and zones from the **Layup Modeler** dialog box:

Application	Advanced Simulation
Prerequisite	**Layup Modeler** dialog box open with a ply defined and selected
Location in dialog box	**Draping Input**

Draped material orientation

What is it?

When you create a layup, the fiber orientation of one of the draped plies (**Ply 1** by default) defines the **Material Orientation** for the surface. The **Material Orientation** in turn defines the zero degree direction for all the plies on the surface. You can select any ply assigned to a surface as the ply that defines the **Material Orientation** for the surface.

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 **Material Orientation** node and select the ply.

Laminate Optimization

What is it?

New optimization capabilities have been added to the **Laminate Modeler** dialog box. In the new **Optimization** group, if you select the **Enable Optimization** check box, you can optimize a characteristic of a laminate physical property to achieve a design objective such as mass, stiffness, or thermal expansion properties. At the same time, you can constrain other similar characteristics. Supported design objectives and constraints are listed below.

The software automatically adjusts specified design variables and calculates the resultant objective values until the design objective is optimized. The software can optimize both continuous design variables such as an orientation angle, and discrete design variables such as the existence of a ply or a material.

Defining a design objective and constraints

An **Objective** is the laminate characteristic you hope to improve by optimizing the laminate. Define one or more objectives for the optimization, together with a **Rule** for the objective (**Minimization** or **Maximization**). If you click **Config** in the new **Optimization** group, you can use options in the **Laminate Optimizer Configuration** dialog box to define both design objectives and design constraints.

You can define any of the following laminate characteristics as an **Objective**:

- Total Mass
- Buckling Factor
- X Young's Modulus
- Y Young's Modulus
- XY Shear Modulus
- XY Poisson's Ratio
- X Thermal Expansion Coefficient
- Y Thermal Expansion Coefficient
- XY Thermal Expansion Coefficient

A **Constraint** is a ply characteristic that you want to constrain to previously defined values during the optimization. You can constrain any of the following ply characteristics:

- Total Mass
- Buckling Factor
- Failure Index
- Natural Frequency
- Plies Contiguity
- X Young's Modulus
- Y Young's Modulus
- XY Shear Modulus
- XY Poisson's Ratio
- X Thermal Expansion Coefficient
- Y Thermal Expansion Coefficient
- XY Thermal Expansion Coefficient

Note

The software automatically adds a symmetry constraint for symmetrical stacking recipes.

Specifying design variables

Design variables are the ply characteristic(s) you want the optimizer to adjust to achieve the objective. In the **Design Variable Manager** group, you define a design variable and add it to the list of available design variables. From this list, you can assign it to a selected ply. The design variables are:

- Ply Angle
- Ply Material
- Ply Thickness
- Ply Existence (select the **Removable Ply** check box)

Running the optimization

After configuring the optimization, you click **Launch** to run the optimizer. You can select one of the five optimized laminate definitions to replace the original laminate.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A previously defined laminate physical property or a laminate currently defined in the **Laminate Modeler** dialog box.
Simulation Navigator	For a previously defined laminate physical property, right-click the mesh collector® **Edit**® **Modify Selected**
Location in dialog box	**Optimization** group **Design Variable Manager** group **Assign Design Variable** group

Graphical post processing

What is it?

Laminate Graphical Post Report reads one or more NX results sets and graphically displays envelopes of ply stresses, strains, failure indices and margins of safety. Because the envelopes show the worst case ply results over all the selected solutions, subcases and iterations you do not need to individually review the load case results. **Laminate Graphical Post Reporting** uses the laminate properties in your current FEM, so that you can switch failure theories and even modify your laminates after the solution to perform “what-if” analyses. This means that you do not have to resolve your solution to understand the effect of changing failure theories or even changing the laminate itself.

Laminate Graphical Post Reporting creates a laminates metasolution. The software stores the metasolution data as a binary universal file.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A simulation file open with a laminate model in a Nastran, ANSYS or ABAQUS solution
Toolbar	**Laminates** ® **Graphical Post Report**
Menu	**Insert**® **Laminate**® **Graphical Post Report**

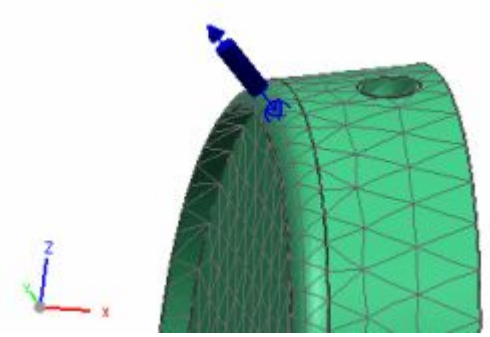
Response Simulation

Placement of sensor normal to element faces

What is it?

The Response Simulation **Sensor** dialog box now includes a **Normal** orientation option. Use this option to easily create the sensor normal to the element faces associated with the node on which you place the sensor. You can also reverse the normal direction.

The **Normal** option works only if you place the sensor on a node on the free face of an element. To determine the normal for the node, the software creates a virtual reference face using the nodes from the surrounding element free faces.



Sensor placed normal to a virtual reference face

For more information, see Sensors overview in the Advanced Simulation help.

Why should I use it?

This option is especially useful in areas of the model where the normal direction is not easy to specify using the direction components (for example, on edge blends or fillets).

Where do I find it?

Application	Advanced Simulation
Toolbar	**Response Simulation**® **Sensors and Strain Gages** ® **New Sensor**
Simulation Navigator	Right-click the **Sensors** node® **New Sensor**
Location in dialog box	**Sensor** dialog box® set **Direction** to **Normal**

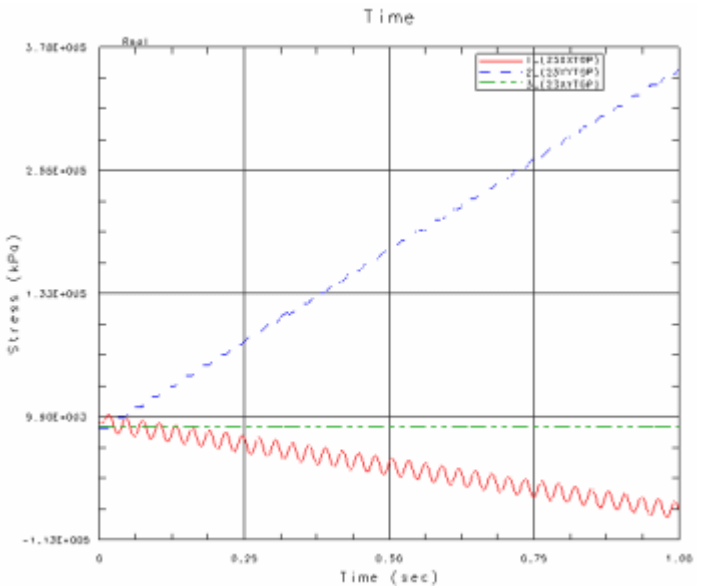
Strain gage

What is it?

The strain gage is a new tool, similar to the sensor, that lets you specify a nodal or elemental location on the model at which to evaluate stress or strain results in a specified direction. Strain gages define:

- Location
- Coordinate system for the stress or strain results
- Components of the stress or strain results

After defining strain gages, you can create an analysis event and excitation(s) and then perform an evaluation for selected strain gages on the model. The **Evaluate Strain Gages** command generates strain or stress response results for selected data components (for example, XX, YY, XY, Von Mises, or the legs of the strain gage).



Sample stress results

To represent some of the most widely used foil strain gages, NX offers these four types:

- **Uni-axial**
- **Bi-axial**
- **0, 45, 90 Rosette**
- **0, 60, 120 Rosette**

For more information, see Strain Gages overview in the Advanced Simulation help.

Why should I use it?

You can use a strain gage object to simulate the location and orientation of a real foil-type strain gage on your model.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Response Simulation® New Strain Gage**
Simulation Navigator	Right-click the **Strain Gages** node® **New Strain Gage**

NX Nastran physical damping

What is it?

In your Response Simulation evaluations, you can now include the physical (also called non-modal) viscous and/or hysteretic damping. Physical damping is based on physical and material properties in the NX Nastran model. NX Nastran calculates this damping as part of the normal modes solve.

To use physical damping, you must first define it in your model before solving for the normal modes.

Viscous damping:

- CDAMPi element — Define the **Viscous Damping** value in the PDAMP physical property table.
- CDAMP2 element — Define the **Viscous Damping** value in the mesh associated data for the element.
- CVISC element — Define the **Viscous Damping** values in the PVISC physical property table.
- CBUSH element — Define the **Viscous** values in the PBUSH physical property table.

Hysteretic (structural) damping:

- CELAS and CELAS2 elements — Define the **Damping Coefficient** in the PELAS physical property table.
- CBUSH element — Define the **Structural** value in the PBUSH physical property table.

- G parameter — Create a **Solver Parameters** modeling object in the **Parameters** tab in the **Edit Solution** dialog box.
- Material damping — Define the damping coefficient in the materials record.

After you solve for the normal modes and create the Response Simulation solution process, use the **Physical Damping Settings** dialog box to enable the physical damping in your response evaluations.

You can view the percentage of the damping ratio for each mode in the **Response Simulation Details View** in the **%Phys Visc** and **%Phys Hyst** columns. The values displayed are the diagonal terms of the viscous damping matrix or hysteretic damping matrix multiplied by any Scale factor you defined.

For more information, see Damping overview in the Advanced Simulation Help.

Where do I find it?

Application	Advanced Simulation
Simulation Navigator	Right-click the **Normal Modes** node® **Physical Damping Settings**

Velocity Impact excitation

What is it?

In Response Simulation, you can simulate a drop test or constant velocity impact test using the new **Velocity Impact** excitation.

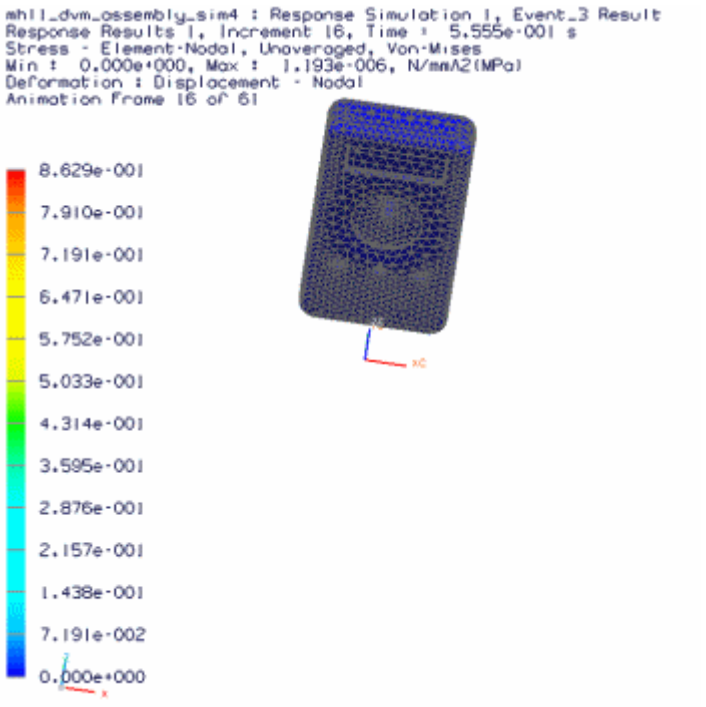
You can define this excitation at a single nodal location on your model in a transient event. The software uses a velocity enforced motion to simulate the impact, so you must define the impact node on a solved enforced motion location.

Two impact methods are available:

- **Drop Impact** — Lets you specify either the drop height or the desired velocity at the time of impact. The software calculates the velocity from the specified drop height or vice versa, depending on which of the two values you specify.
- **Constant Velocity Impact** — Lets you specify a constant velocity to be used for the impact.

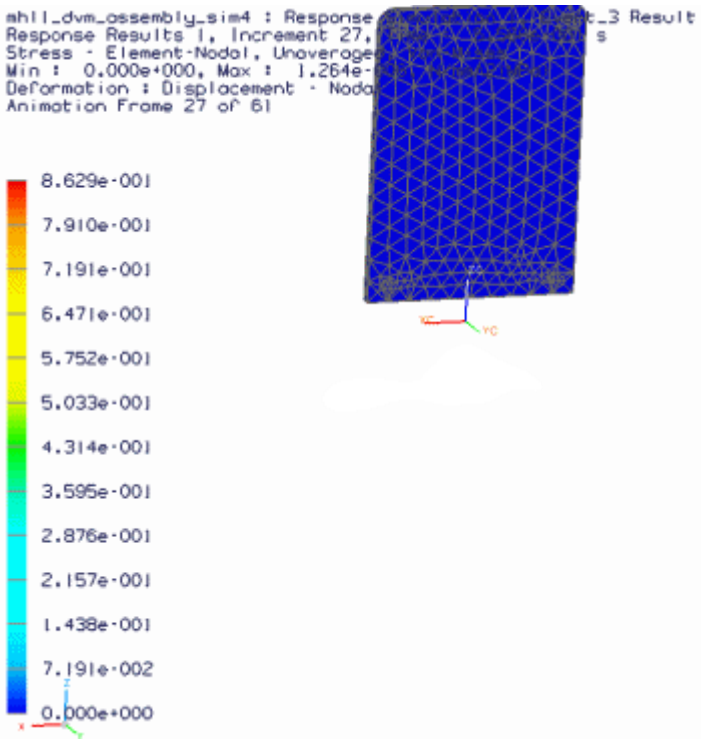
The following example shows the animation of a drop impact. The handheld electronics device is dropped on the front-right corner (a node on the

front-right corner is specified as the enforced motion location). The first animation shows the Von mises stress over the entire model.



Handheld electronics device (Von mises stress)

The second animation shows the stress distributed to the internal circuit board, which is the area of interest. The front and back covers of the device are hidden.



Circuit board inside device (Von mises stress)

For more information, see About velocity impact excitations in the Advanced Simulation help.

Why should I use it?

You can use this excitation to simulate a controlled drop test or a constant velocity impact test for which you know the length of the impact pulse or the height of the drop. This excitation type is not intended for analyzing localized stress at the impact point. It cannot predict nonlinear response, but can provide good approximations of response away from the impact point where response may still be linear (such as in the internal circuit board in the second animation).

Where do I find it?

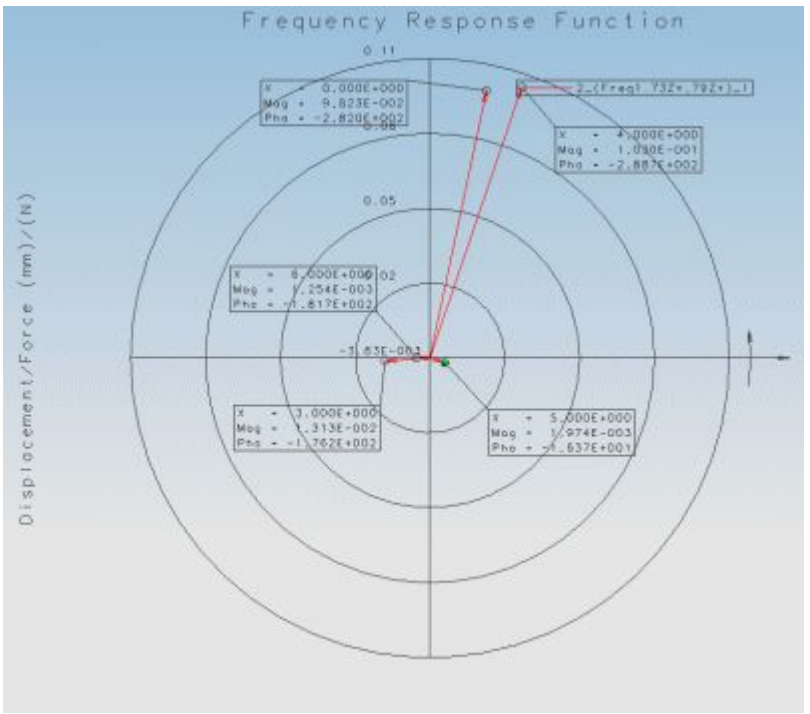
Application	Advanced Simulation
Prerequisite	Transient event
Toolbar	**Response Simulation® New Velocity Impact Excitation**
Simulation Navigator	Right-click the **Excitations** node® **New Excitation® Velocity Impact**

Modal Contribution

What is it?

The **Modal Contribution** option available with the **Evaluate FRF** and **Evaluate Transmissibility** commands has been enhanced. Now you can use **Modal Contribution** to determine the individual normal mode frequencies that contribute most to the overall amplitude and phase at multiple selected input frequencies. The software generates a response function for each input frequency and output node or element you select.

The contributions of the selected input frequencies are displayed in a polar plot, which displays the complex numbers in a circular grid representing amplitude and phase angle. Each arrow on the polar plot represents a normal mode frequency (the X value is the mode ID) and output node or element on the model.



For more information, see About Modal Contribution in the Advanced Simulation help.

Where do I find it?

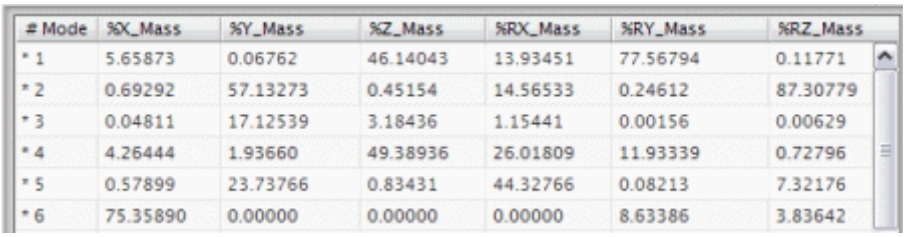
Application	Advanced Simulation
Toolbar	**Response Simulation® Evaluate FRF** or **Evaluate Transmissibility**
Menu	**Tools® Response Simulation® Evaluate Transfer Functions® FRF** or **Transmissibility**

Simulation Navigator	Right-click the response simulation node® **Evaluate FRF** or **Evaluate Transmissibility**
Location in dialog box	**Evaluate FRF** or **Evaluate Transmissibility** dialog box® set **Property Method** to **Modal Contribution**

Rotational effective mass in Simulation Navigator

What is it?

You can now display the percentage of the total effective mass contributed by a selected mode in one of the three rotational directions (RX, RY, and RZ). The mass values appear in the **Response Simulation Details View** in the **Simulation Navigator** along with the X, Y, and Z effective mass values.



# Mode	%X_Mass	%Y_Mass	%Z_Mass	%RX_Mass	%RY_Mass	%RZ_Mass
* 1	5.65873	0.06762	46.14043	13.93451	77.56794	0.11771
* 2	0.69292	57.13273	0.45154	14.56533	0.24612	87.30779
* 3	0.04811	17.12539	3.18436	1.15441	0.00156	0.00629
* 4	4.26444	1.93660	49.38936	26.01809	11.93339	0.72796
* 5	0.57899	23.73766	0.83431	44.32766	0.08213	7.32176
* 6	75.35890	0.00000	0.00000	0.00000	8.63386	3.83642

The new mass columns are hidden by default; you must enable them in the navigator.

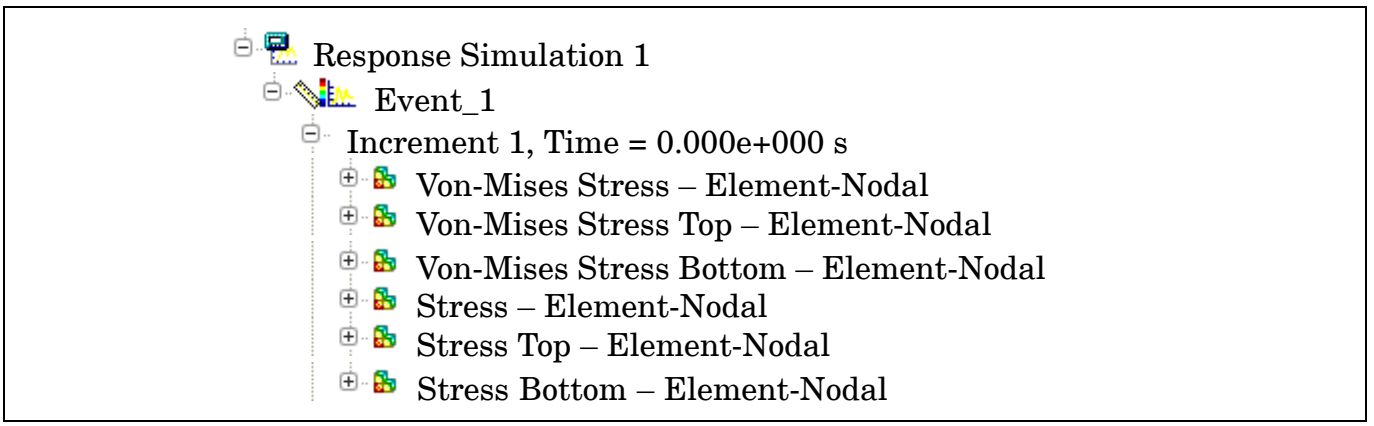
Where do I find it?

Application	Advanced Simulation
Simulation Navigator	**Response Simulation Details View**® right-click in the column heading® **Columns**® **RX_Mass**, **RY_Mass**, and **RZ_Mass**

Post-processing results organization

What is it?

The response contour results for Response Simulation are now displayed using standard Advanced Simulation post-processing. All results now appear in the **Post-Processing Navigator**, where you can use the full array of analysis commands to control their display.



Sample response results in Post-Processing Navigator

Also, in the **Simulation Navigator**, when you use the **Quick View** command on the **Modal Representation** node or on the **Contour Results** node, a **Post View** is created and you can use the **Post-Processing** toolbar to control the display.

For more information about response contour results, see Dynamic responses overview in the Advanced Simulation help.

Where do I find it?

Application	Advanced Simulation **Simulation Navigator**® right-click **Normal Modes**, **Constraint Modes**, or **Attachment Modes** node® **Quick View**
Simulation Navigator	**Contour Results** node® right-click result node® **Quick View**

Cross-spectral density response function

What is it?

You can correlate the response between two selected points on your model using the new **Evaluate CSD Function** command in a Random event. You can specify a “reference” node or element and multiple “response” nodes or elements. The software creates a cross-spectral density function that correlates the response between the two points, with result data types such as displacement, velocity, acceleration, and so on.

Response functions are stored as function records in an AFU file with the same name as your Response Simulation solution process.

For more information, see CSD response functions overview in the Advanced Simulation help.

Where do I find it?

Application	Advanced Simulation
Prerequisite	Random event
Toolbar	**Response Simulation**® **Evaluate Function Response** ® **Evaluate CSD Function**
Menu	**Tools**® **Response Simulation**® **Evaluate Functions**® **Evaluate CSD Function**
Simulation Navigator	Right-click the Event node® **Evaluate Function** **Response**® **CSD**

Quasi-Static analysis events

What is it?

A Quasi-Static analysis event type is now available. With a Quasi-Static event, you can evaluate the static response of your model to one or more simultaneous time-varying static excitations. The excitation for a Quasi-Static event must consist of distributed attachment modes generated from loads in the Dynamics subcase for an SEMODES 103 – Response Simulation solution.

In the response simulation, you scale the excitation using a unitless scalar time function. The software calculates the response at each instant in time by linearly combining (superposing) the distributed attachment modes. Only the model's stiffness is considered in the calculation; time integration is not used. These simpler calculations allow the software to process the event quickly, especially when it includes a large number of modes.

For more information, see About quasi-static events in the Advanced Simulation help.

Why should I use it?

This event type is useful if you are interested only in static results and need faster solution performance than a full dynamic solution.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A load in the Dynamics subcase for an SEMODES 103 – Response Simulation solution
Toolbar	**Response Simulation**® **New Event**
Simulation Navigator	Right-click **Response Simulation** solution process node® **New Event**

SRS/Time function conversion

What is it?

Two new function conversion commands have been added to the **Function Tools for Response Simulation** utility:

SRS→Time	Lets you convert shock response spectrum (SRS) functions to time functions.
Time→SRS	Lets you convert time functions to SRS functions.

For more information, see Excitation functions overview in the Advanced Simulation help.

Why should I use it?

Suppose that SRS excitations are given as the specifications for your analysis, but you want to perform a Transient analysis. You can first use the **SRS→Time** conversion command to generate time functions according to the given SRS specifications. Likewise, you can use the **Time→SRS** command to generate an SRS specification from a given time function.

Where do I find it?

Application	Advanced Simulation
Toolbar	**Response Simulation® Function Tools for Response Simulations**

Motion Simulation

Mechatronics and control

NX and Simulink co-simulation

What is it?

You can use the new co-simulation feature to simulate your motion mechanism connected to a control system diagram modeled in MATLAB® Simulink® from The Mathworks™.

Co-simulation is an integrated solve that runs in both NX Motion Simulation and Simulink, enabling closed-loop system analysis. During co-simulation, the result outputs from the mechanical system and the control system design tools are transferred at a constant sampling time. The control system receives information about the state of the mechanism (such as displacements, velocities, or accelerations) and then responds with instructions that provide input to a joint driver, force, or torque load in the mechanism.

For more information, see Mechatronics co-simulation overview in the Motion Simulation help.

Why should I use it?

The co-simulation feature lets the mechanism designer and the control system designer evaluate the interaction between the mechanical and logic systems early in the design process. This early testing helps verify whether the control system design is robust enough to control the nonlinear dynamic mechanism. At any stage in the design process, users can simulate multiple variants of the control system, choose the logic, adjust the gain, and apply the latest mechanical data.

Where do I find it?

Application	Motion Simulation
Prerequisites	• **Co-Simulation** selected in the **Environment** settings • **Control/Dynamics** solution • Location of MATLAB executable specified in **Customer Defaults** (**Analysis**® **RecurDyn** tab) • Location of Simulink control system diagram (*.mdl file) specified in solution attributes (**Solution** dialog box® **Cosim** group)
Toolbar	**Motion**® **Plant Input** and **Plant Output**
Menu	**Insert**® **Plant**
Motion Navigator	Right-click the Motion Simulation file® **New Plant**

PMDC Motor, Signal Chart, and Motor Driver

What is it?

You can now represent the variable timing of a permanent magnetic direct current (PMDC) motor using the new PMDC motor object combined with a signal chart and motor driver. This ability lets you define motors controlled by open-loop or closed-loop systems.

PMDC Motor

Use the **PMDC Motor** object to define the electrical parameters of the motor, including nominal voltage, impedance, inductance, resistance, and initial current. These parameters define a velocity profile that the motor must follow.

For more information about motors, see PMDC motor overview in the Motion Simulation help.

Signal Chart

The signal chart provides the input signal to the motor. Typically, the input signal will be 0 (motor stopped), -1 (motor rotating in opposite direction of the motor driver), or 1 (motor rotating in the direction of the motor driver). The software multiplies the input signal by the **Nominal Voltage** parameter of the motor to arrive at the applied voltage, which determines the speed of the motor.

There are two types of signal chart: **Open Loop** and **Closed Loop**.

- **Open Loop** — Controls the motor using a **Timing Chart** function. The motor follows the defined function and does not react to changes in the system. You can specify a different signal at each time interval to control the motor velocity.

- **Closed Loop** — Controls the motor using one or more sensors and event conditions so the software can respond to system changes. The sensors monitor a simulation value (Displacement, Force, Velocity, or Acceleration) between two markers or in a joint. The signal chart reads the data from the sensors and applies the event condition (less than or greater than a threshold value).

For more information about sensors, see Sensor.

For more information about signal charts, see Signal chart overview in the Motion Simulation help.

Motor driver

The Motor driver associates a PMDC motor with a signal chart and imparts the resulting motion to a revolute joint or cylindrical joint.

For steps to create a motor driver, see Create motor driver in the Motion Simulation help.

Where do I find it?

PMDC Motor

Application	Motion Simulation
Prerequisite	**Motor Driver** selected in **Environment** settings
Toolbar	**Motion® PMDC Motor**
Menu	**Insert® Control® Motor® PMDC Motor**
Motion Navigator	Right-click the motion simulation® **New Motor**

Signal Chart

Application	Motion Simulation

Prerequisite	**Motor Driver** selected in **Environment** settings
Toolbar	**Motion® Signal Chart**
Menu	**Insert® Control® Signal Chart**
Motion Navigator	Right-click the motion simulation® **New Signal Chart**

Motor driver

Application	Motion Simulation
Prerequisite	• **Motor Driver** selected in **Environment** settings • **Control/Dynamics** solution • Revolute or cylindrical joint
Toolbar	**Motion® Driver**
Menu	**Insert® Driver**
Motion Navigator	Right-click the motion simulation® **New Driver**

Timing Chart function

Application	Motion Simulation
Shortcut menu	**XY Function Navigator® NX AFU Files** node® right-click AFU file® **Create**
Location in dialog box	**XY Function Editor** dialog box® set **Purpose** to **Motion®** set **Function Type** to **Timing Chart**

General functionality

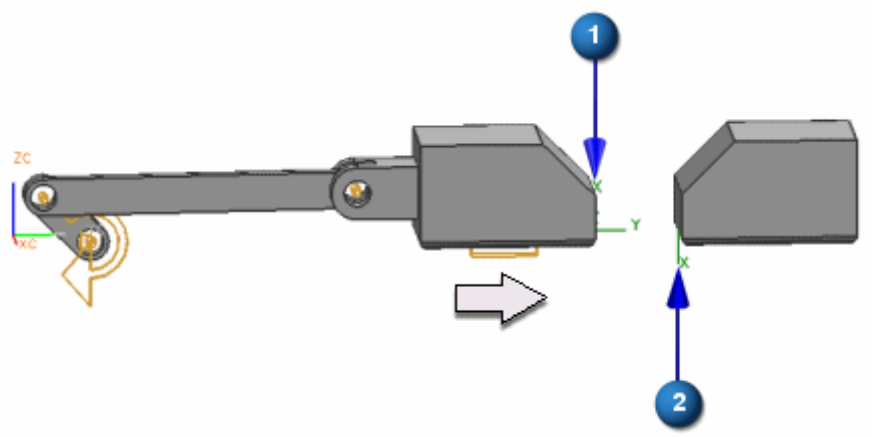
Sensor

You can now use a sensor to monitor the positions of motion objects related to simulation conditions such as displacement, velocity, acceleration, and force.

You can define a sensor with an absolute reference frame on one marker or one joint, or a relative sensor between two markers.

For example, with a displacement sensor, you can monitor the linear distance between two markers. When the markers come within the specified distance, the sensor is triggered and the signal chart sends a new signal to the motor driver.

In the picture below, a sensor monitors relative displacement between marker and marker . You could define a signal chart using this sensor to reverse the motor before the moving body comes within a specified distance of the stationary body.



For more information, see Sensors overview in the Motion Simulation help.

Why should I use it?

Typical uses for sensors include:

- Capturing data about the mechanism for use in a co-simulation.
- Controlling the voltage signal sent to an electric motor (using a series of sensors with a closed-loop signal chart). For more information about motors and signal charts, see PMDC Motor, Signal Chart, and Motor Driver.
- Pre-defining a specific output of interest for your mechanical model that is often used. For example, plotting a relative rotation around an individual X, Y, or Z axis.

Where do I find it?

Application	Motion Simulation
Toolbar	**Motion®** **Sensor**
Menu	**Insert® Sensor**
Motion Navigator	Right-click the motion simulation® **New Sensor**

Nonlinear springs, dampers, and bushings

What is it?

You can now define nonlinear behavior of spring, damper, and bushing Motion objects using the new **Stiffness and Damping** function type. This function lets you create a table of force or torque values that are proportional to displacement or velocity.

When defining the Motion object, choose **Spline** as the stiffness or coefficient **Type**. This opens the **XY Function Editor**, where you can create a **Stiffness**

and Damping table function. You then enter a table of values for the range of the force.

For more information, see About nonlinear springs, dampers, and bushings in the Motion Simulation help.

Why should I use it?

Mechanical components such as rubber mountings, spiral springs, gas springs, and shock absorbers (dampers) show some nonlinearity in real life. The stiffness and damping are not constant, but rather depend on relative displacement or velocity of the components. This new feature lets you more accurately simulate these nonlinear motion objects.

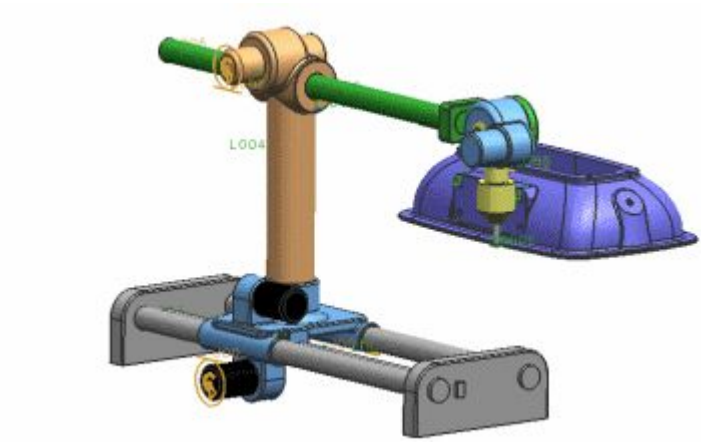
Where do I find it?

Application	Motion Simulation
Toolbar	**Motion® Spring** , **Damper** , or **Bushing**
Location in dialog box	**Type** list® **Spline**

Motion driver for cylindrical joints

What is it?

In a **Normal Run** solution, you can now assign a motion driver to cylindrical joints. In the motion driver definition, you can specify the rotation and translation values independently.



Cylindrical joint driven in rotation and translation

Note

You cannot use this driver with the **Articulation** or **Spreadsheet Run** solution types.

For more information, see Motion driver overview in the Motion Simulation help.

Where do I find it?

Application	Motion Simulation
Toolbar	**Motion® Driver** or **Joint**

Show Intersection Curve Interference type

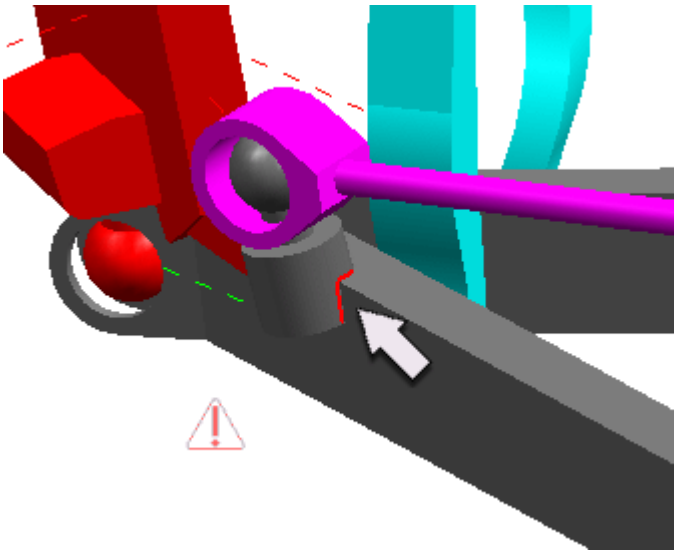
What is it?

The new **Show Intersection Curve** command lets you view the volume of interference between two sets of solid bodies or components during an animation or articulation.

When an interference occurs, a temporary set of curves appears in the model that outlines the volume of the interference. After you exit the animation or articulation, the temporary curves disappear.

Note

This command works only when the interference objects are of the same type; that is, they must be either all solid bodies or all components.

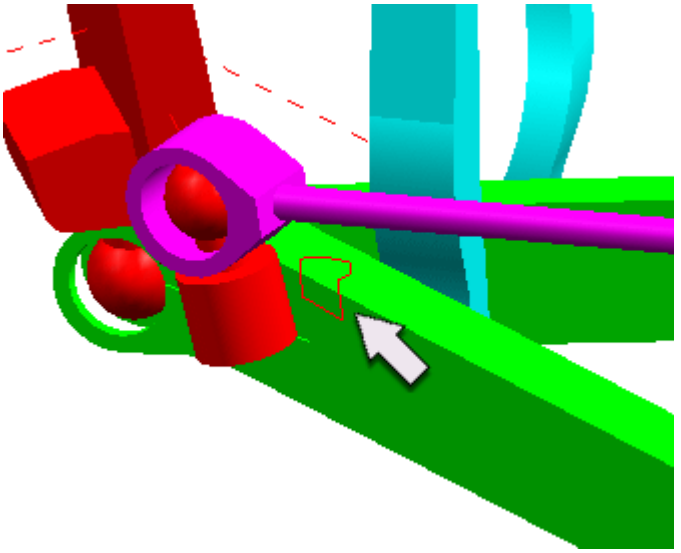


Interference between the lower control arm and ball/cylinder

A warning icon appears in the graphics window to alert you to the interference condition.

During an animation, you can pause at any time step and click **Trace Intersection Curves** in the **Animation** dialog box to outline the

interference at that time step. When you use the **Trace Intersection Curves** command, the interference curves are retained in your model.



Traced interference curves

For more information, see Interference overview in the Motion Simulation help.

Why should I use it?

Show Intersection Curve is ideal for cases in which you want more precise identification of the interference than simple highlighting of the objects, but do not necessarily want to create permanent new objects in your model.

Where do I find it?

Application	Motion Simulation
Toolbar	**Motion® Interference**
Menu	**Tools® Packaging® Interference**
Motion Navigator	Right-click the Motion Simulation® **New Interference**

Usability improvements

Open a Motion simulation file directly

What is it?

You can now open a Motion simulation (*.sim) file directly without first opening the master part. In previous versions, you were required to first open the master part, start the Motion Simulation application, and then open the simulation file.

The geometry associated with the Motion Simulation is loaded automatically, depending on the option you choose in the **Assembly Load Options** dialog box. For example, for faster loading with a large assembly, use the **Structure Only** load option.

Where do I find it?

Application	Motion Simulation
Menu	**File**® **Open**® **Files of Type=Simulation Files (*.sim)**® select a .sim file to open

Link Geometries information category

What is it?

You can view information about a link by right-clicking the link in the **Motion Navigator** and choosing **Information**. The **Information** window displays data in categories such as Link Name, Part ID, Joint Names, Center of Mass, CSYS For Link Inertias, and so on.

This link information includes a new category named **Link Geometries**. The **Link Geometries** category includes links defined on components, curves, solid bodies, sheet bodies, facet bodies, and points. It is listed at the bottom of the **Information** window as shown in the following example.

```
Link Geometries
              Solid Body ID 1236    from part   boom.prt
              Solid Body ID 965     from part   stick.prt
              Component  CYL3       from part   cyl3.prt
```

For more information about links, see Link overview in the Motion Simulation help.

Where do I find it?

Application	Motion Simulation
Motion Navigator	Right-click a link® **Information**

Show the assembly component associated with a link

What is it?

You can view the assembly component associated with a particular link using the new **Expand to Selected** command. In the **Assembly Navigator**, this command expands and highlights the assembly component on which the link is based.

Where do I find it?

Application	Motion Simulation

Prerequisite	• **Component-based Simulation** selected in the **Environment** dialog box • **Assembly Navigator** opened as a separate window
Motion Navigator	Select one or more links® right-click the selection® **Expand to Selected**

Link selection filter in Customer Defaults

What is it?

In the **Customer Defaults** dialog box, you can now control the default selection filter for selecting geometry to create a link. You can set the default filter to:

- **Component**
- **Curve**
- **Solid Body**
- **Sheet Body**
- **Faceted Body**
- **Point**

If you change the selection filter while creating a link, that filter overrides the default filter and becomes the new default filter. However, if you reset the dialog box settings, the selection filter from the **Customer Defaults** dialog box becomes the default filter again.

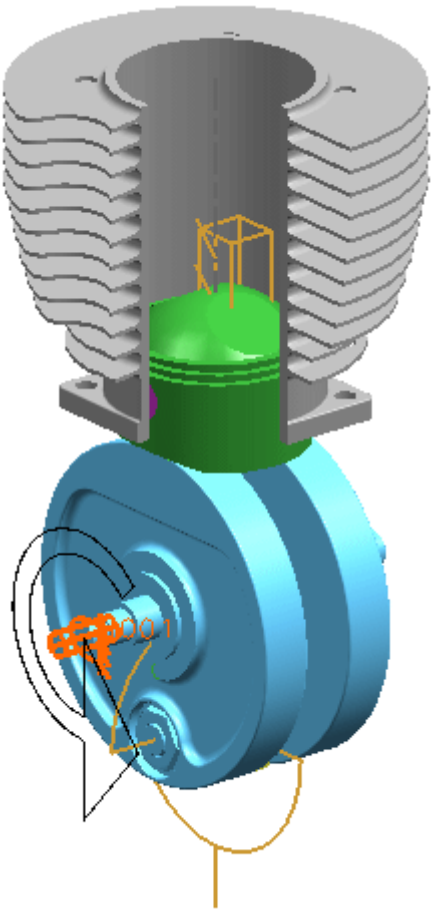
Where do I find it?

Application	Motion Simulation
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Motion® Pre Processor® Link Selection** tab

Stand-alone Driver preview arrow

What is it?

When you create a stand-alone joint driver using the **Driver** dialog box, a preview arrow now indicates the direction of the driven joint as you define the joint. Previously, this direction arrow did not appear until after the joint was created.



Preview arrow indicating direction of the joint driver

For more information about joint drivers, see Motion driver overview in the Motion Simulation help.

Where do I find it?

Application	Motion Simulation
Toolbar	**Motion® Driver**
Menu	**Insert® Driver**
Motion Navigator	Right-click the Motion Simulation® **New Driver**
Location in dialog box	**Preview Direction** group

Functions and Graphing

Persistent graph display settings

What is it?

Now, when you make changes to a graph's display properties, all subsequent functions you plot use these display changes for the duration of the NX session, or until another template is applied. Display changes are saved in memory regardless of whether you save the changes as a template. You can still control display settings with templates as in previous versions.

Also, three new right-click commands have been added to the template nodes in the **XY Function Navigator**:

- **Reload** — Restores the original template settings (the settings that are stored in the template XML file) and discards any display customizations currently stored in memory.
- **Save** — Saves to the XML template all display customizations currently stored in memory. This command is not available with the default template.
- **Reset** — Reverts the default template settings to the original software settings (if you changed the default template).

For more information, see Edit display properties of graph objects in the Functions and Graphing help.

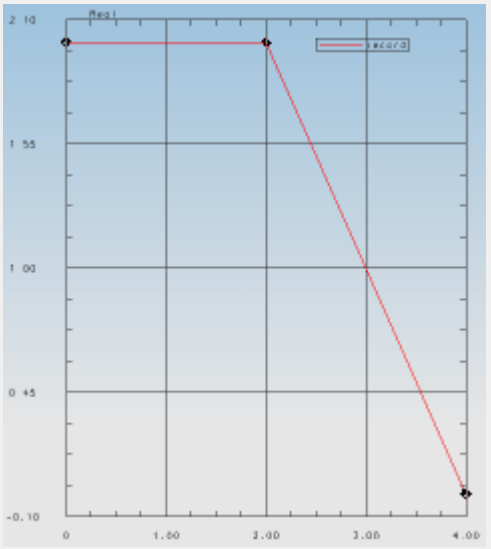
Synchronize functions

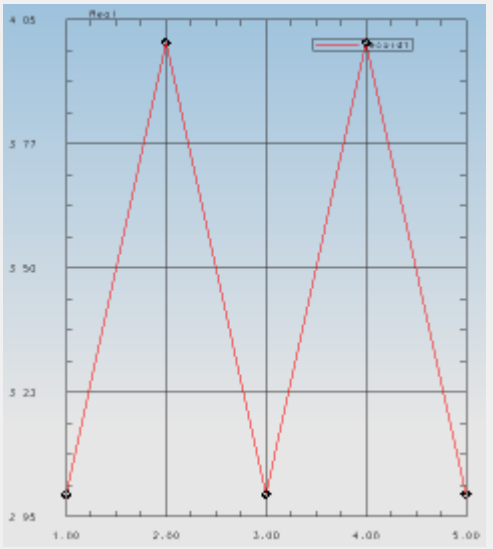
What is it?

Using the **Synchronize** command, you can now ensure that multiple functions all have the same starting point, increment, and number of data points.

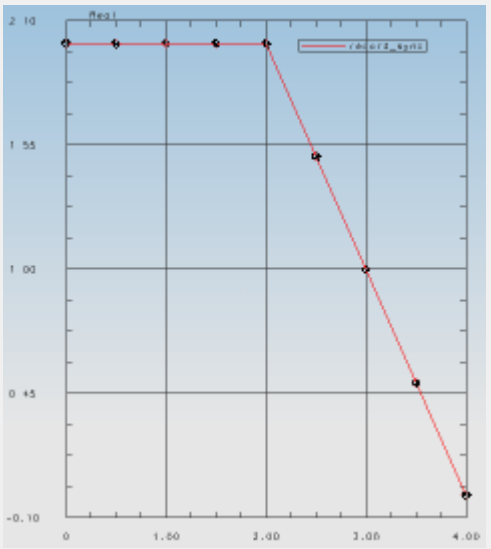
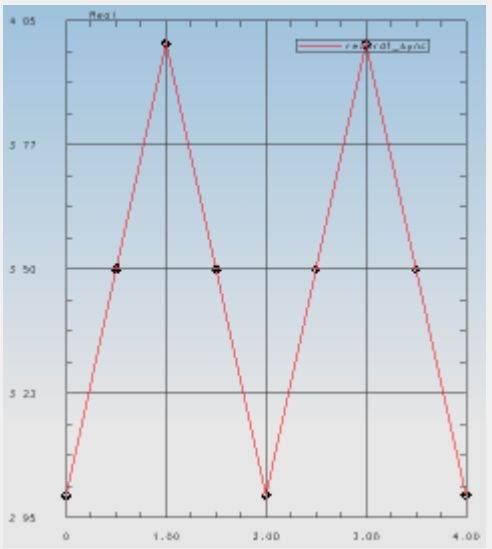
When you synchronize functions, you specify a new increment value. You can also shift the abscissa starting point of one function to align with the starting point of a second function using the **Shift Abscissa to Align Start Point** option. Two new functions are created as shown in the second set of graphs in the following example.

Function 1 (original)Increment=2; Start=0.0; Number of points=3	**Function 2 (original)Increment=1; Start=1.0; Number of points=5**
0.0, 2.0	1.0, 3.0
2.0, 2.0	2.0, 4.0
4.0, 0.0	3.0, 3.0
	4.0, 4.0
	5.0, 3.0





Function 1 (synchronized)Increment=0.5; Start=0.0; Number of points=9	**Function 2 (synchronized)Increment=0.5; Start=0.0; Number of points=9**
0.0, 2.0	0.0, 3.0
0.5, 2.0	0.5, 3.5
1.0, 2.0	1.0, 4.0
1.5, 2.0	1.5, 3.5
2.0, 2.0	2.0, 3.0
2.5, 1.5	2.5, 3.5
3.0, 1.0	3.0, 4.0
3.5, 0.5	3.5, 3.5
4.0, 0.0	4.0, 3.0

Function 1 (synchronized)Increment=0.5; Start=0.0; Number of points=9	Function 2 (synchronized)Increment=0.5; Start=0.0; Number of points=9
	

When you synchronize the functions, the software interpolates them to Even spacing and creates the new, synchronized functions in the specified AFU file.

For more information, see Synchronize functions overview in the Functions and Graphing help.

Why should I use it?

Before you can apply multiple functions as excitations in a Response Simulation analysis event, you must ensure the functions all have the same starting point, increment, and number of data points.

Where do I find it?

Application	Advanced Simulation or Motion Simulation
Toolbar	**Function Math Operations® Synchronize**
Menu	**Tools® Math Operations for Functions® Basic Math® Synchronize**

New function types

What is it?

Two new function types have been added to support new NX Motion Simulation features.

- **Timing Chart** — Lets you simulate the variable timing of an electric motor with a table of velocity values and time intervals. For more information, see PMDC Motor, Signal Chart, and Motor Driver.

- **Stiffness and Damping** — Lets you simulate the nonlinear behavior of springs, dampers, and bushings with a table of force and displacement values. For more information, see Nonlinear springs, dampers, and bushings.

Where do I find it?

Application	Motion Simulation
XY Function Navigator	**NX AFU Files** node® right-click AFU file® **Create**
Location in dialog box	**XY Function Editor** dialog box® set **Purpose** to **Motion**® set **Function Type** to **Timing Chart** or **Stiffness and Damping**

Chapter

11 Optimization and Analysis

NX Analysis

One-step Formability Analysis

What is it?

The **One-step Formability Analysis** command now has improved intermediate unform stability and enhanced solver performance to solve a large number of mesh element cases.

This command now enables you to:

- Specify the target region and unform region from a different sheet body.
- Customize and save specific related properties for **One-step Formability Analysis** material in the standard NX material library.
- Examine the mesh quality and view a report of the mesh elements status.
- Input three match points to define spring back constraints.
- Output spring back facet bodies.
- Save the inputs and settings in the part file. The data is retrieved when you reopen the part file and run the **One-step Formability Analysis** command.

Why should I use it?

Use this command for FEM based Sheet Metal forming analysis. You can use the command to perform complete or intermediate unforming, flatten a Sheet Metal part, or calculate thinning, stress, strain and springback to predict the risk of forming.

Where do I find it?

Application	Gateway
Menu	**Analysis® One-step Formability Analysis**

Application Toolbar	Modeling **Body Design® One-step Formability Analysis**
Menu	**Die Engineering® One-step Formability Analysis** **Tools® Vehicle Design Automation® Body Design® One-step Formability Analysis** **Tools® Vehicle Manufacturing Automation® Die Engineering® One-step Formability Analysis**
Application Toolbar	Progressive Die Wizard **Sheet Metal Tools® One-step Formability Analysis**

Optimization and Sensitivity Study

Optimization

What is it?

The new **Optimization** dialog box replaces the **Optimization Wizard**. It is easy to use and has the following new options.

- **Algorithm**: You can directly select an algorithm from the **Algorithm Type** list. The new Optimization function includes two new local optimization algorithms and four new global optimization algorithms.

  For more information, see Knowledge Fusion optimization enhancement.

  You can also add your own user function-based algorithms to the new NX Optimization tool.

- **Algorithm Details**:

  The **Local algorithms** (Powell and Conjugate Gradient) are generally better for finding a precise answer when you are already quite confident that you know generally where the desired answer should be found. The answer returned by a local optimization will be highly dependent on the starting point of the analysis, and may *miss* a better answer located in a different part of the design space.

  – **Powell**: Minimizes each design variable in sequence, rotating through the list of all design variables until the minimum is reached. This method does not use derivatives, and is quite intuitive to watch.

  – **Conjugate Gradient**: If we can calculate derivatives easily, then this method converges very quickly. However, in most CAD cases the

objective will not have a mathematical form and we use a difference method to approximate derivatives. **Powell** is usually recommended over **Conjugate Gradient**.

The global algorithms (**Global Simplex**, **Simulated Annealing**, **Lexicographic**, and **Pattern Swarm**) perform some searching of the specified design space for good alternatives before converging on the best option. As such, they generally perform more design iterations than the local algorithms, but are generally much better at finding the best answer even when several relatively good alternatives are present in the design space.

- **Global Simplex**: Uses a random sampling to find likely areas where the global minimum may exist, and then employs a geometric (simplex) method to achieve the minimum. This method does not rely on derivative information and is rather intuitive.

- **Simulated Annealing**: Uses a *slow cooling* method to reach a least energy state (much like crystallization). This method is time-consuming but is known to provide global solution in a majority of cases.

- **Lexicographic**: was originally designed to be used with multiple objectives. Objectives are ranked in order of importance, and the optimum solution is found by using an annealing method to minimize the objective functions starting with the most important and proceeding in order of importance. (Note: Multiple objectives have not yet been exposed in NX.)

- **Pattern Swarm**: Uses the newest (2006) algorithm for global optimization which does not use derivatives. In most optimization methods we start out with a single point in the design space and choose the direction to take the next step to optimization. This method starts out selecting a set of points, and then tries to move each into the optimal position. The process is time consuming but does an impressive job of converging to the global solution very reliably.

You can specify detailed settings for the selected algorithm. Depending on the algorithm selected, different **Converging Speed** options are available to choose from.

Why should I use it?

The **Optimization** command enables you to set up, run, and monitor shape optimization on NX models. Use the new options to select the required algorithm type and specify detailed settings for the selected algorithm.

Where do I find it?

Application	Modeling
Prerequisite	Your role must be set to **Advanced**, **Advanced with Full Menus**, or an equivalent custom role.
Menu	**Analysis**® **Optimization and Sensitivity**® **Optimization**
Location in dialog box	**Algorithm** group® **Algorithm Type** **Settings** group® **Algorithm Details**

Sensitivity Study

What is it?

Sensitivity Study is a new command that lets you perform a systematic design study as part of the design process in an attempt to understand the effect of design variables on desired outcomes. Users can vary one or more model parameters over specified ranges to observe the effect on the model's performance and to evaluate design trade-offs.

A Sensitivity Study is set up and run much like an Optimization Study, with one fundamental difference. Whereas an optimization attempts to find one single *best* solution, the Sensitivity Study generates all cases in the design space, and lets you browse the results of the study to examine the entire design space, or to look at the effect of specific design variables on a desired outcome in different areas of the design space.

You can specify Design Variables using any of the following distribution methods:

- Uniform distribution: Evenly-spaced values over a specified range.
- Normal distribution: Random values selected from a normal distribution located within a specific range.
- Gamma distribution: Random values selected from a gamma distribution located between zero and a specified upper limit.

Why should I use it?

The **Sensitivity Study** command enables you to easily generate a range of models in batch mode and understand more complex parameter interactions, without repetitive manual updating.

You can use Sensitivity Study results in robustness analysis by identifying design alternatives that are less susceptible to variation in the design inputs.

Where do I find it?

Application	Modeling

Prerequisite	Your role must be set to **Advanced**, **Advanced with Full Menus**, or an equivalent custom role.
Menu	**Analysis® Optimization and Sensitivity® Sensitivity Study**

Chapter

12 Product Validation

Check Requirements

Save check result to Teamcenter

What is it?

The new **Save Result to Teamcenter** right-click option allows Requirement Check results to be saved to the Teamcenter database.

Why should I use it?

NX Requirement Check results that have been published to Teamcenter can then be used in Teamcenter workflows in conjunction with the Validation Workflow handlers to ensure that product requirements are being met before parts are released.

Where do I find it?

Application	Check Requirements
Resource bar	NX Part Navigator

Check-Mate

Check-Mate Treat Warning as Pass

What is it?

Check-Mate now includes a **Treat Warning as Pass** option. If this option is on, it treats the warning error level as a pass level that generates a Check Flag. You can also use "–treat_warning_as_fail" as a switch option for the ug_check_part command line utility.

Where do I find it?

Application	Check-Mate
menu	**Analysis→Check-Mate→Run Options**.
Location in dialog box	**Run Options** page® **Smart Check options**

Command line utility	ug_check_part

Checker and function enhancements

What is it?

Enhancements to existing checkers and functions let you check dimension styles and get product interface information.

Why should I use it?

This enhancement widens the abilty to check drafting functionality.

Where do I find it?

Application	Check-Mate
Location	**Analysis→Check-Mate→Tests** dialog box® **Categories** list box® **Get Information→Product Interface**

Sheet Metal Validation Checkers

What is it?

Check-Mate now provides sheet metal validation checks. The new checks support sheet metal work flows and forming.

Why should I use it?

These types of checker routines benefit the majority of sheet metal customers.

Where do I find it?

Application	Check-Mate
Toolbar	Check-Mate® Run Tests
Menu	**Analysis→Check-Mate→Run Tests**
Location in dialog box	Tests page® **Categories**

Chapter

13 Manufacturing Design

Mold and Die Tools

Mold Design

Initialize Project — Select attributes

What is it?

You can create a list of user-defined attributes in the *custom_attr_template.xls (xs4)* file, and use that list to add attributes to the top level assembly file (**_top.prt*) as you initialize a mold assembly. This list appears in the **Project Initialize** dialog box in the **Attributes** group.

You can edit the *custom_attr_template.xls (xs4)* file by clicking the new **Customized Attributes** button in the **Settings** group.

Why should I use it?

You can define attributes to:

- Support a bill of materials (BOM).
- Identify tooling assembly components to support automation regardless of the actual file name.
- Initialize attributes for drafting.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Initialize Project**

Initialize Project — File template structure

What is it?

There are three mold tooling assembly templates:

- **Mold.V1**
- **ESI**
- **Original**

Mold.V1

The **Mold.V1** assembly:

- Is the recommended structure.
- Contains a new **parting-set** subassembly to let you change the mold coordinate system after you define the parting.

  Note

  The **parting-set** subassembly contains the parting, molding, shrink, and original product parts.

- Has more robust parting capability; you never need to identify a seed face in the cavity or core.
- Supports multiple region definition for slides and lifers.
- Supports merged cores and cavities for family molds or molds with multiple instances.
- Supports face colors inherited from regions you define with the **Extract Regions and Parting Lines** command.

ESI

ESI is an acronym in English for *early supplier involvement*. The **ESI** structure:

- Supports your evaluation of moldability.
- Lets you review product model changes.
- Supports testing the **Swap Model** command in a small assembly with much faster updates.

Original

- The **Original** configuration has the same structure as the **Default** structure had in recent releases.

- Use it if you prefer the previous structure.

For the full assembly structures of all three configurations, please see the online Help file for Configuration templates.

Mold.V1 product template structure		
prod		
	workpiece	
	parting-set	
		shrink
		molding
		parting
	core	
	cavity	
	trim	
	prod_side_a	
	prod_side_b	

After initialization, the parting set contains the original product, as shown in the following figure.

Mold.V1 parting-set structure	
parting-set	
	shrink
	molding
	parting
	original product part

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Initialize Project**
Menu	**Tools® Process Specific® Mold Wizard® Initialize Project**
Location in dialog box	**Project Setting** group® **Configuration** list® **Mold.V1**

New parts created by rename

What is it?

You can use the new default rename method instead of the clone method to create new component parts when you use:

- The **Mold Base Management** dialog box.
- Any command that uses the Standard Part Management system.

You can change the default method using the **Part Installation Method** customer defaults option.

Why should I use it?

The rename method is faster than the clone method, and it does not automatically save new parts.

When you close new parts without saving them, you no longer have to remove unwanted cloned files from your project folder.

Where do I find it?

Commands

Application	Mold Wizard
Toolbar	**Mold Wizard**
Menu	**Tools® Process Specific® Mold Wizard**
Commands supported	**Initialize Project** **Slider/Lifter Design** **Standard Part Management** **Sub-Insert Design** **Mold Base Management** **Cooling Component Design** **Workpiece Insert Design** **Electrode Design** **Concept Design**
Location in dialog box	File parameters and file commands sub window

Part Installation Method customer default

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Mold Wizard®** General page

Concept Position

What is it?

You can select the **Concept Design** check box in the **Standard Part Management** or **Mold Base Management** dialog box to open the **Concept Position** dialog box when you are ready to position a part.

Points are used to represent components. The points have attributes to define the catalog, name, location, and all parameters of the standard part you select.

When you select the **Concept Design** option in the Mold Base Management system, or in any of the Standard Part Management system dialog boxes, the **Concept Position** dialog box opens instead of the part placement dialog box. You can:

- Specify a mounting plane.
- Specify a point.
- Select a circular curve or edge to specify its center point.

Why should I use it?

You can use the new **Concept Position** dialog box to:

- Simplify the mold design process and mold assembly structure by using conceptual design.
- Design the layout of a mold base and standard components very quickly by postponing adding the actual parts.
- Install all of your pre-defined parts in one step by using the **Concept Design** command.

For example, you can customize a mold base template assembly to include concept design points and planes for specific ejector pins and other standard parts. After you add your custom mold base template to your mold tooling assembly, you can add the pre-configured standard parts in one step using the new **Concept Design** dialog box.

Where do I find it?

Application	Mold Wizard
Prerequisite	In the **Position** list, the selected method must be **POINT** or **PLANE** and the **Associative Position** check box must not be selected.

Commands supported	**Initialize Project**	**Slider/Lifter Design**
	Standard Part Management	**Sub-Insert Design**
	Mold Base Management	**Cooling Component Design**
	Workpiece Insert Design	**Electrode Design**
Location in dialog box	File parameters and file commands sub window	

Concept Design

What is it?

In the **Concept Design** dialog box, you can:

- Select concept design points and planes you created using the **Concept Position** dialog box.
- Add all of the parts described by the points and planes you select in one operation.

Why should I use it?

Use this command to:

- Simplify your mold assembly.
- Design mold tooling layouts quickly.

If you use the same set of standard parts in all mold assemblies of a given type, you can avoid repeating the same installation steps for every standard part, by installing all the common standard parts using a template part that contains concept design points.

Where do I find it?

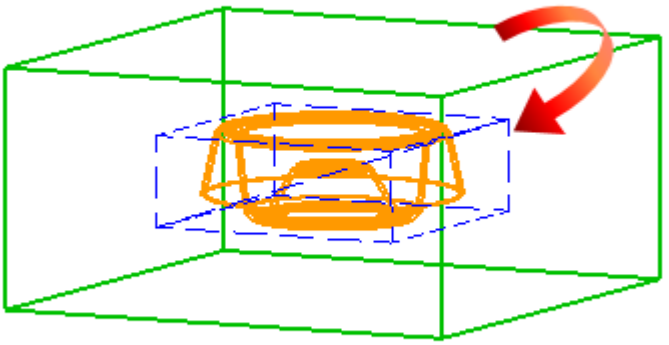
Application	Mold Wizard
Toolbar	**Mold Wizard® Concept Design**
Menu	**Tools® Process Specific® Mold Wizard® Concept Design**

Workpiece

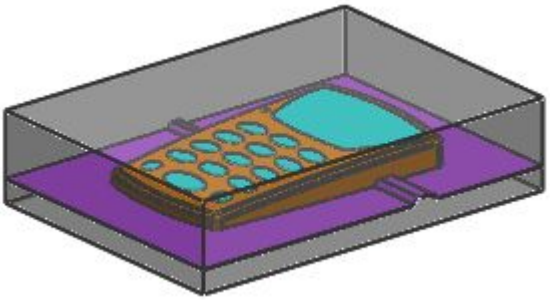
What is it?

You can now:

- Design single workpiece instances or combined workpieces.
- Define a **User Defined Block** workpiece that is sketch-based.
- Display the bounding box of the original product model.



- Use a translucent workpiece, in the **Mold.V1** template configuration.



Why should I use it?

Use the sketch-based, extruded workpiece when you want to:

- Control blends and chamfers.
- Create a custom workpiece section to extrude.
- Design a workpiece for a merged core, cavity, or both.

You can display the product bounding box when you want to visualize the relationship between the extremities of the original product body and the current workpiece body.

Where do I find it?

Application	Mold Wizard
Prerequisite	The sketch-based, extruded workpiece is available only in the **Mold.V1** configuration, which you can select during product initialization.
Toolbar	**Mold Wizard® Workpiece**
Menu	**Tools® Process Specific® Mold Wizard® Workpiece Settings** group® **Show Product Bounding Box**
Location in dialog box	**Workpiece Method** group, **Workpiece Method® User Defined Block**.

Cavity Layout

What is it?

You can now use the **Vector** dialog box to specify the first direction in automatic layout.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Cavity Layout**
Menu	**Tools® Process Specific® Mold Wizard® Cavity Layout**
Location in dialog box	**Layout Type® Specify Vector**

Static Interference Check

What is it?

You can now check a mold tooling assembly for possible interferences between its components.

The Static Interference Check is synchronized with the **Assembly Clearance Analysis** analysis in assemblies. See the *Assemblies* online Help for more information.

Why should I use it?

A static interference check helps you improve your design by showing you where any interferences or missed pockets occur.

You can:

- Synchronize results with the **Assembly Clearance Analysis** command.
- Store the analysis and results for future rechecking.

Where do I find it?

Application	**Mold Wizard**
Toolbar	**Mold Tools® Interference Check**
Menu	**Tools® Process Specific® Mold Wizard® Mold Tools® Interference Check**

Stock Size

What is it?

The new **Stock Size** button opens the **Edit Stock Size** dialog box from the **Mold Tools** toolbar. This means that you can now assign a stock size to a mold part without having to open the **BOM Record Edit** dialog box.

You can still edit existing stock sizes with options in the shortcut menu and the **Settings** group of the **BOM Record Edit** dialog box.

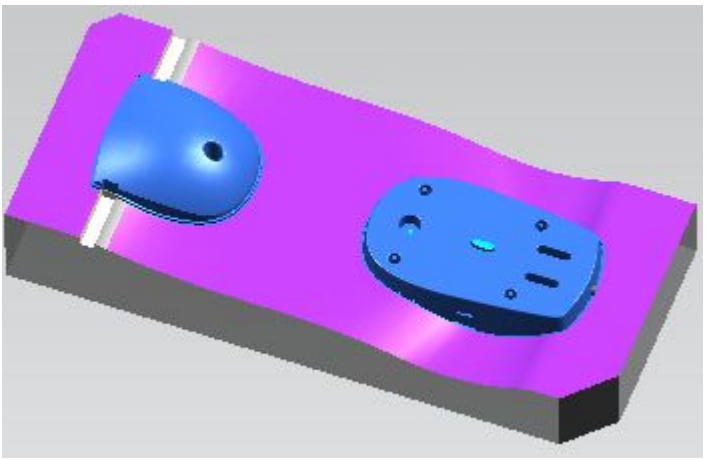
Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Mold Tools® Stock Size**
Menu	**Tools® Process Specific® Mold Wizard® Mold Tools® Stock Size**

Merge Cavities

What is it?

You can design inserts consisting of multiple workpieces. You have the option to unite several bodies to combine them, or to subtract several workpiece bodies from an overall insert body that encompasses all the individual volumes.



Why should I use it?

You can link cavity instances to a combined workpiece, core, or cavity part to:

- Create a single workpiece for multiple product instances, or for multiple products.
- Accommodate features that are physically different between cavity instances, for example, corners or cooling channels.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Mold Tools® Merge Cavities**
Menu	**Tools® Process Specific® Mold Wizard® Mold Tools® Merge Cavities**

Mold CSYS

What is it?

The **Mold CSYS** command is enhanced to take advantage of a new assembly structure in the new **Mold.V1** configuration. The product assembly structure of the **Mold.V1** configuration contains a new **parting-set** subassembly.

Mold.V1 parting-set structure
parting-set
shrink
molding
parting
original product part

Why should I use it?

When you adjust the mold coordinate system, the **parting-set** subassembly maintains the relationship between the original product model and the linked molding, shrink, and parting bodies. This eliminates reported cases where some parting sheets fail to update if the mold coordinate system is changed after the parting is defined.

Where do I find it?

Application	Mold Wizard
Prerequisite	You must use the new template structure **Mold.V1** to take advantage of this enhancement.
Toolbar	**Mold Wizard® Mold CSYS**
Menu	**Tools® Process Specific® Mold Wizard® Mold CSYS**

Parting — Multiple Regions

What is it?

You can now use the following set of commands in the **Parting Manager** dialog box to define multiple regions and to create geometry for each region:

- **Extract Regions and Parting Lines**
- **Edit Parting Lines**
- **Guide Line Design**
- **Create/Edit Parting Surfaces**
- **Create Cavity and Core**

For each region, you can define and organize:

- Parting surfaces

- Parting lines
- Patch surfaces
- Patch solids

Why should I use it?

You use multiple regions to:

- Define bodies for slide and lifter heads.
- Diagnose problems for complex parting, by:
  - Defining several regions.
  - Highlighting the sheets for each region and sewing them manually.
  - Identifying sewing problems for each region.
  - Testing each region for trimming a body.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Parting**
Menu	**Tools® Process Specific® Mold Wizard® Parting**

Parting — Define Regions

What is it?

The new **Define Regions** dialog box has all of the important capabilities of the different legacy methods.

When you click **Extract Regions and Parting Lines** in the **Parting Manager** dialog box, the **Define Regions** dialog box opens. You can:

- Examine a tree diagram showing counts of undefined, core, and cavity faces.
- Define as many additional regions as you require, for example, to extrude slide heads.
- Select individual region faces or search for region faces.
- Specify the color or translucency of a selected region.
- Define region names.

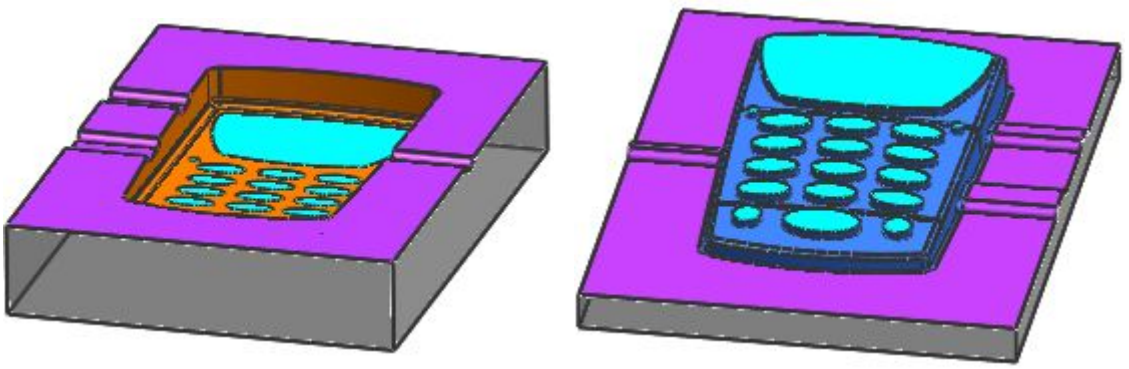
Note

Region names are used as component file names. The new components contain a linked solid body for a particular region.

- Select options to create region sheets, parting curves, or both.

The region features, parting curves, and guide lines are displayed in and selectable from the tree diagram of the **Parting Manager** dialog box.

Region name	Face count	Layer
All faces	58	
Undefined faces	0	
Cavity region	27	28
Core region	31	27
New region	0	29



Why should I use it?

Multiple regions are very useful in slide design. Regions are used to create core, cavity, and slide solids in the **Define Cavity and Core** command.

When you create core and cavity sheets and parting curves, you no longer have to select a method to define regions.

Core and cavity faces are detected and constructed automatically, based on the draw vector. You can continue to define as many regions as you need for such things as slides and lifters.

You can search for region faces based on a seed face and boundary curves. Existing parting edges are automatically identified as boundary objects for a new region. This reduces the amount of work to select faces for a new region.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Parting**
Menu	**Tools® Process Specific® Mold Wizard® Parting**

Parting — Guide Line Design

What is it?

You can use the **Guide Line Design** command to create guide lines that:

- Automatically define parting segments.
- Support normal, tangential, snap to WCS axis, and user defined direction options.
- Are used to establish directions for parting sheets and to trim parting sheets.

Why should I use it?

You can use the **Guide Line Design** command to:

- Save time by defining sheet boundaries and sweep or extrude vectors for parting surface creation.
- Group parting lines into different segments; for each segment you can a parting surface of a type you select.

Where do I find it?

Application	Mold Wizard
Prerequisite	Parting curves must be created before guide lines.
Toolbar	**Mold Wizard® Parting® Define Guide Lines and Segments**
Menu	**Tools® Process Specific® Mold Wizard® Parting**

Parting — Create Cavity and Core

What is it?

You can now:

- Create a solid for each region you define.
- Automatically search for parting and patch sheets for each region.

- Map the parting sheet, region sheet, and patch sheet colors to the solids for each region.

  Note

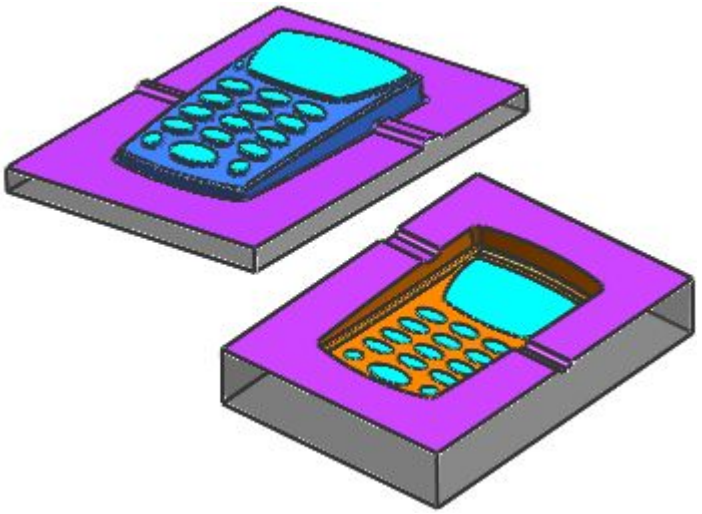
  This feature applies only to the **Mold.V1** configuration template.

- Highlight the parting surface boundary if trim fails.

- Eliminate the need for seed sheet bodies.

  Note

  This feature applies only to the **Mold.V1** configuration template.



Why should I use it?

You can quickly create solid bodies for slide heads.

You can use colors in the parting sheets to specify each manufacturing process by color.

You can quickly diagnose problems when failed parting boundaries are highlighted.

Where do I find it?

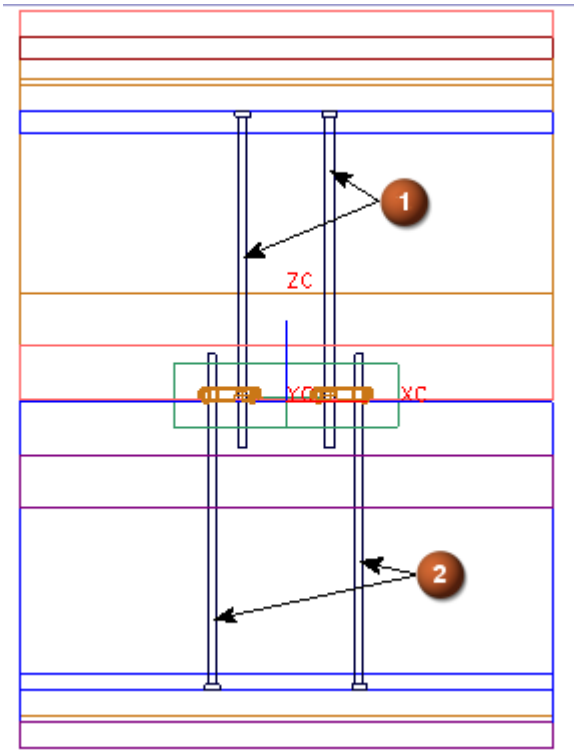
Application	Mold Wizard
Toolbar	**Mold Wizard® Create Cavity and Core**
Menu	**Tools® Process Specific® Mold Wizard® Create Cavity and Core**

Ejector pins

What is it?

You can use Improved ejector pin models, placement methods, and postprocessing to:

- Select cavity side ejector pins for reverse ejection.
- Trim ejectors without having to specify the trim direction.
- Trim ejectors without having to adjust the close fit distance.
- Review all trim errors by clicking , instead of responding to a separate warning message for every error.



(1) Cavity side ejectors
(2) Core side ejectors

Where do I find it?

Add ejector pins

Application	Mold Wizard
Toolbar	**Mold Wizard® Standard Parts**

Menu	**Tools® Process Specific® Mold Wizard® Standard Parts**
Location in dialog box	Available parts list® **Cavity Side Ejector Pin**

Trim ejector pins

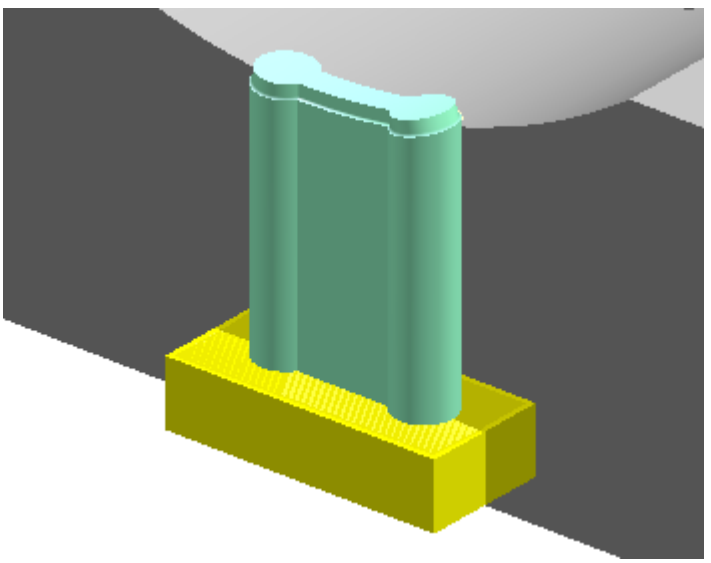
Application	Mold Wizard
Toolbar	**Mold Wizard® Ejector Pin**
Menu	**Tools® Process Specific® Mold Wizard® Ejector Pin**

Design Inserts

What is it?

You can use the **Design Inserts** command on the **Mold Tools** toolbar to:

- Select any body to use as a sub-insert.
- Design a foot shape for the sub-insert, including foot clearance.
- Create stock size information automatically.
- Create body clearance for a cylindrical sub-insert.
- Automatically create a new component containing your sub-insert body design and a standard foot.



Why should I use it?

Use this command when you require a sub-insert body of your own design in place of the sub-insert standard library parts.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Mold Tools® Design Inserts**
Menu	**Tools® Process Specific® Mold Wizard® Mold Tools® Design Inserts**

Pocket Design — Select pocketing body

What is it?

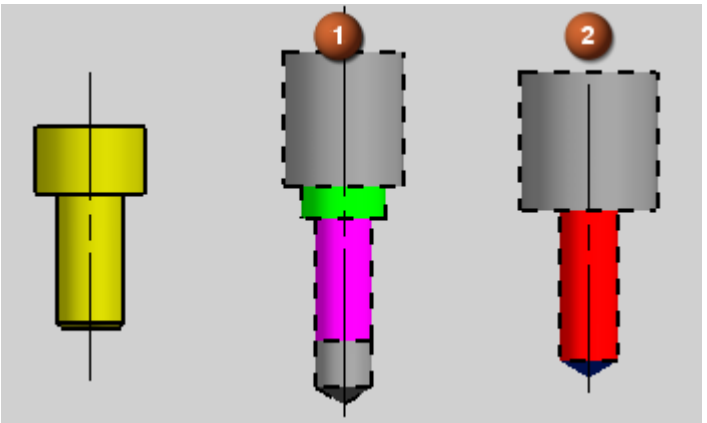
In the **Pocket Design** command, you can now:

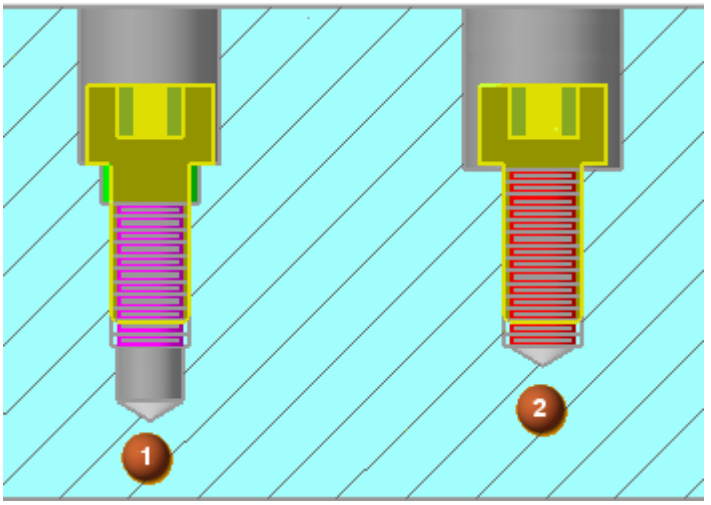
- Select a body from a **FALSE** reference set with multiple bodies.
- Show or hide shortcut buttons for the two choices in the **Select Types** list.

The magenta face in body (1) and the red face in body (2) have the **MW_HOLE_THREAD** attribute set to **1**.

Body (1) is modeled to provide a short clearance distance at the top of the **M8x1.25** threaded hole, represented by the green face. The tap drill diameter in body (1) extends about 10 mm beyond the threaded distance.

Body (2) has a larger counterbore diameter than body (1). The full length of the tap drill diameter of body (2) is threaded.





Why should I use it?

You can model different **FALSE** reference set bodies to provide a choice of pocket shape, counterbores, clearances, and threaded distance.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Pocket Design**
Menu	**Tools® Process Specific® Mold Wizard® Pocket Design**
Location in dialog box	**Tool** group, **Select Types**

Pocket Design — Add material

What is it?

In the **Pocket Design** command, you now have two modes of pocket creation:

You have

- **Subtract Material**
- **Add Material**

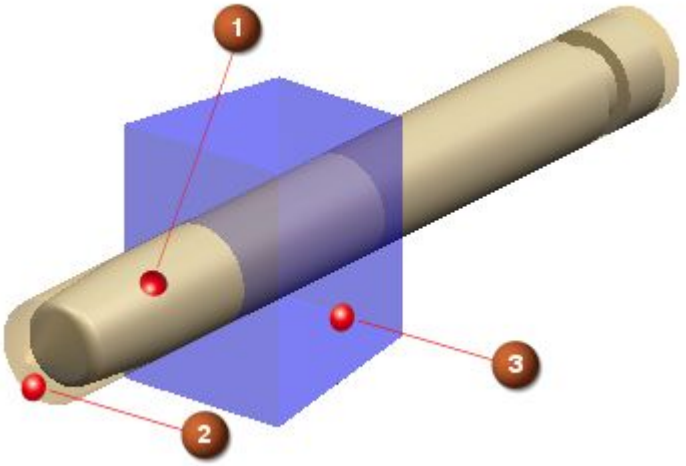
You can display the new **ADD_MATERIAL** reference set to show bodies that represent added material.

The first figure shows a mold lock pin that requires both a subtracted hole shape, and an added tapered pad.

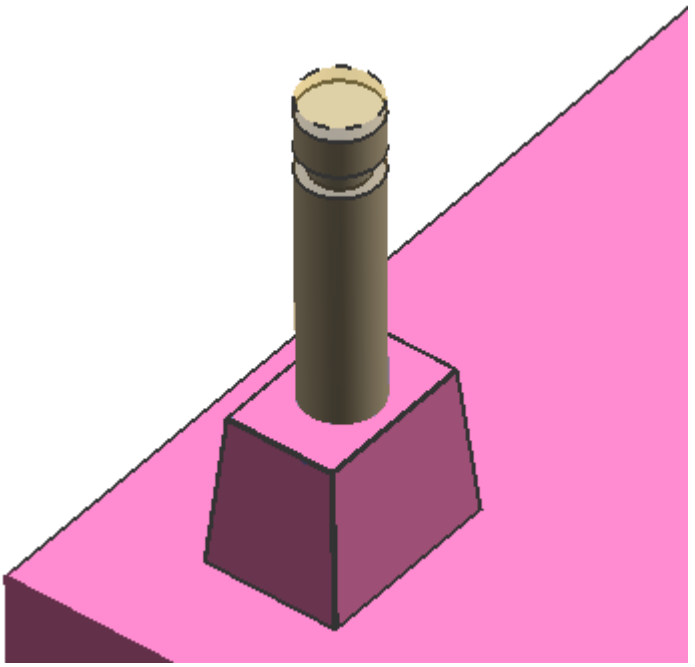
The pin (1) is in the **TRUE** reference set.

The hole body (2) is in the **FALSE** reference set.

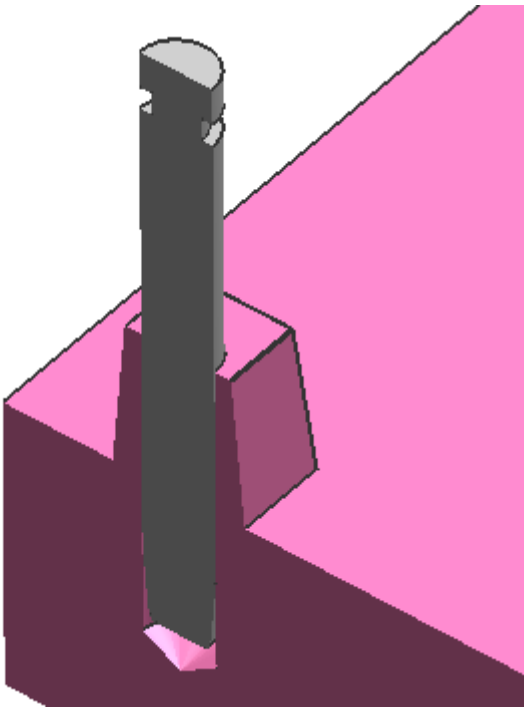
The pad body (3) is in the **ADD_MATERIAL** reference set.



The following figure shows a plate modified by using the **Add Material** pocket mode with the **ADD_MATERIAL** reference set.



The following figure shows the plate after a second pocket operation using the **Subtract Material** mode with the **FALSE** reference set.



Why should I use it?

Use it when you need to add material to a target body to properly represent the contours needed to position a standard part.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Pocket Design**
Menu	**Tools® Process Specific® Mold Wizard® Pocket Design**
Location in dialog box	**Mode® Add Material**, **Reference Sets® ADD_MATERIAL**

Bill of Material

What is it?

In the **BOM Record Edit** dialog box, you can now:

- Have as many columns as you need.
- Sort and edit columns easily.
- Enter up to 132 characters in each string.
- Perform actions from the shortcut menu instead of using buttons.

- Calculate stock sizes for multiple selected components in one step.

Where do I find it?

Application	Mold Wizard
Toolbar	**Mold Wizard® Bill of Material**
Menu	**Tools® Process Specific® Mold Wizard® Bill of Material**

Hole Table

What is it?

The **Hole Report** toolbar has been replaced with the following changes:

- A **Hole Table** dialog box that combines the previous **Create Hole Report** and **Auto Create Notes** functions. This dialog box is opened when you click **Hole Report** on the **Mold Wizard** toolbar.
- The **Edit Hole Description** and **Hole Report Preferences** functions are now customer defaults in the drafting section.

Where do I find it?

Hole Table

Application	Mold Wizard
Toolbar	**Mold Wizard® Hole Table**

Edit Hole Description and **Hole Report Preferences** functions

Application	Mold Wizard
Menu	**File® Utilities® Customer Defaults**

Wire EDM Start Hole

What is it?

You can use the new **Wire EDM Start Hole** command to sketch locations and diameters for wire EDM start holes.

Why should I use it?

Wire EDM start holes make it easier for you to use the Wire EDM commands in the Manufacturing application.

Where do I find it?

Application Toolbar	Mold Wizard **Mold Wizard® Mold Tools ® Wire EDM Start Hole**
Menu	**Tools® Process Specific® Mold Wizard® Mold Tools® Wire EDM Start Hole**

NX Electrode Design

Undersized Body

What is it?

You can use the **Undersized Body** command to create a solid representation of the actual size electrode head required on an EDM. The following orbit types are supported:

- Circular
- Square
- Triangular
- Spherical
- Select Points

The undersized body is an approximation made by the following method:

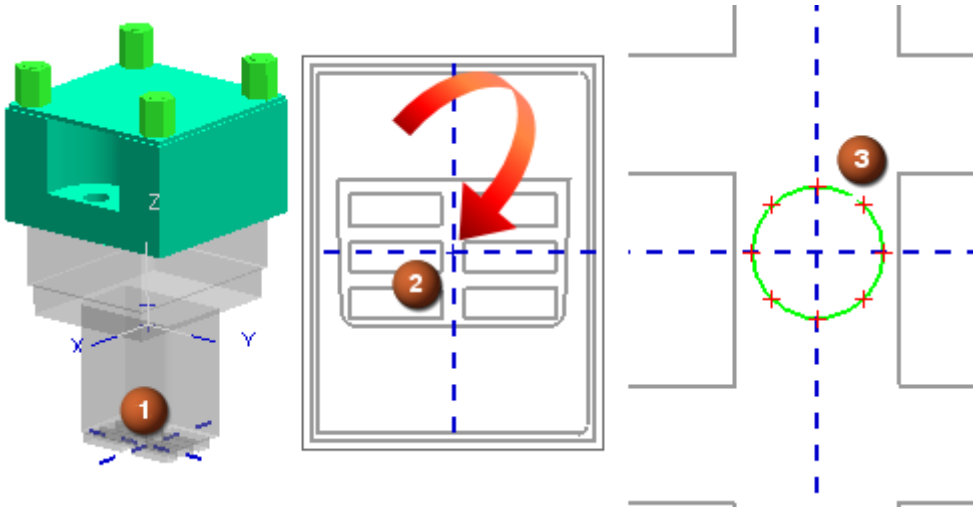
1. In the electrode body, at the electrode reference point or center of the orbit, a circle is calculated. The radius is the radius of the orbit plus the allowance for the spark gap.

2. Points are calculated on the circle:

   - Exactly three points for a triangular orbit.
   - Exactly four points for a square orbit.
   - A number of points determined by the **Divided Angle** setting for a circular orbit.

3. The body is copied to each point.

4. The approximated body is the Boolean intersection of all of the copied bodies.



Reference point (1) in trimetric view

Reference point (2) is origin point for copies

Destination points for copies (3), with **Divided Angle = 45**

Where do I find it?

Application	Electrode Design
Toolbar	**Electrode Design® Undersized Body**
Menu	**Tools® Process Specific® Electrode Design® Undersized Body**

Copy Electrode

What is it?

You can use the **Copy Electrode** command to create electrode component copies to burn the same shape in a different area. There are two options:

- **Transform**
- **Mirror**

Why should I use it?

If you have identical geometry in several places, you need no longer construct a separate electrode body for each location.

Where do I find it?

Application	Electrode Design
Toolbar	**Electrode Design® Copy Electrode**

Menu	**Tools® Process Specific® Electrode Design® Copy Electrode**

Progressive Die Design

Static Interference Check

What is it?

The **Static Interference Check** command is redesigned to synchronize with the **Assembly Clearance** analysis functions. You can now check a progressive die assembly for possible interferences among its components.

Note

A static interference check is similar to a **Check Clearances** analysis in assemblies. See the *Assemblies* Help for more information.

Why should I use it?

A static interference check helps you improve your design, because it shows you where any interferences or missed pockets or relief occur.

Where do I find it?

Application Toolbar	Progressive Die Wizard **Progressive Die Wizard® Tooling Validation** **® Static Interference Check**
Menu	**Tools® Process Specific® Progressive Die Wizard® Validation Tools® Static Interference Check**

Multiple die plates

What is it?

You can now split multiple die plates at the same time.

Note

All the die plates must be the same length. Also, multiple die plates are split at the same location.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Die Base**

Menu	**Tools® Process Specific® Progressive Die Wizard® Die Base**
Location in dialog box	**Die Base Management®** Design Tools page® **Single Plate**

Wire EDM start hole design

What is it?

You can now create Wire EDM start holes easily with the new **Wire EDM Start Hole** function.

Why should I use it?

You can generate Wire EDM start holes automatically to make it easier for you to use the Wire EDM functions.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Progressive Die Tools** **® Wire EDM Start Hole**
Menu	**Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® Wire EDM Start Hole**

Force calculation reuse

What is it?

You can now calculate the piercing force and save the result as an attribute with your scraps. The piercing force is propagated to the piercing punch as a part attribute.

Why should I use it?

If you save the force calculation result, you can use it in downstream design and in CAE strength analysis operations.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Force Calculation**
Menu	**Tools® Process Specific® Progressive Die Wizard® Force Calculation**

Piercing

What is it?

The **Piercing** function has the following enhancements:

- A new **Rename Dialog** check box, which lets you rename user-defined punch and slug holes
- A choice of user-defined punch heads (**Screwed Head**, **Welded Head**, and **Pad Head**)
- The ability to design a slug hole for cases where the die base has two bottom plates
- The ability to easily create holes in freeform surfaces using the **Freeform Scrap** and **Die Insert** options
- The ability of **Fine Blanking** to support multiple spreadsheets. You can also choose a predefined spreadsheet for die-punch clearance.

If your linked scrap has any changes, click **Update Piercing Insert**. You can also link a piercing insert to a different scrap using **Shape Association**.

Where do I find it?

Application Toolbar	Progressive Die Wizard **Progressive Die Wizard® Piercing**
Menu	**Tools® Process Specific® Progressive Die Wizard® Piercing**

Direct Unfolding

What is it?

The **Direct Unfolding** function has the following enhancements:

- You can directly edit the neutral factor and the developed length in the **Bend List** of the **Direct Unfolding** dialog box, regardless of whether the part is bent or unbent.
- You can define multiple pre-bends for a bend whose angle is greater than 180 degrees.
- You can define or edit pre-bends even after using the **Convert to Sheet Metal** function.

- You can create associative intermediate stages for a direct unfolding operation. You can assign names and station numbers to these intermediate stages.

- When a bend is not recognized by the **Convert to Sheet Metal** function, a symbol (*) appears before its name for easy distinction.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Sheet Metal Tools** **® Direct Unfolding**
Menu	**Tools® Process Specific® Progressive Die Wizard® Sheet Metal Tools® Direct Unfolding**

Bend Operation

What is it?

The **Unbend**, **Rebend**, and **Overbend** options are moved to the new **Bend Operation** dialog box from the **Direct Unfolding** dialog box. This reduces the complexity of the direct unfolding operation.

You can use the **Bend Operation** command only on parts that you converted to sheet metal in the **Direct Unfolding** operation, or on sheet metal parts you created in the NX Sheet Metal application.

Why should I use it?

Bend Operation lets you unbend, rebend, and overbend converted bends without entering the NX Sheet Metal application and without needing a reference face or edge selection.

Where do I find it?

Application	Progressive Die Wizard
Prerequisite	**Bend Operation** can only be used on parts that are converted to sheet metal.
Toolbar	**Progressive Die Wizard® Sheet Metal Tools® Bend Operation**
Menu	**Tools® Process Specific® Progressive Die Wizard® Sheet Metal Tools® Bend Operation**

Strip Layout

What is it?

The design process for the **Strip Layout** function is streamlined, and the function now supports dragging.

A new dockable **Strip Layout** navigator replaces the previous **Strip Layout** dialog box and its pages.

Many of the functions that were previously available in the dialog box are now located on shortcut menus for the navigator nodes.

Why should I use it?

The navigator tree makes the strip layout information available in one location, instead of scattering the information across several dialog box pages. This makes it easier for you to view and understand the relationships.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Strip Layout**
Menu	**Tools® Process Specific® Progressive Die Wizard® Strip Layout**

Current parent

What is it?

You can now specify that the current parent should be the default parent when you add new designs in, for example, the **Standard Parts**, **Piercing Insert Design**, and **Insert Group** functions.

Why should I use it?

When several designers work on a project at the same time under different subassemblies, the **Current Parent** customer default specifies the default parent for each designer's setup.

Where do I find it?

Application	Progressive Die Wizard
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Progressive Die Wizard® Standard Parts**

Initialize Project

What is it?

The **Initialize Project** function now supports multiple project templates.

Why should I use it?

You can preconfigure several project templates for concurrent design or to reuse an existing design.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Initialize Project**
Menu	**Tools® Process Specific® Progressive Die Wizard® Initialize Project**

WAVE Control

What is it?

You can now use **WAVE Control** to control your progressive die project's WAVE link updates.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Progressive Die Tools** **® WAVE Control**
Menu	**Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® WAVE Control**

Molded Part Variation

What is it?

You can now use **Molded Part Variation** to check sheet metal part thicknesses and locate sharp corners in progressive die projects.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Progressive Die Tools** **® Molded Part Variation**
Menu	**Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® Molded Part Variation**

Scrap Design

What is it?

The **Scrap Design** dialog box has the following enhancements:

- The sketch function is embedded.
- You can assign a **Station Number** for scrap. Downstream, the **Strip Layout** function automatically lays the scrap with your specified station number.
- You can use the **Apply Minimum Radius** option on scraps with sharp corners.
- The **Overlap** and **Overcut** functions are now associated with the scrap.
- A new overcut type, **Normal**, can be used for most design needs.
- A new **Grouping** type, which lets you assign colors to scraps with the same area, and provides an option that automatically finds scraps with the same area.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Scrap Design**
Menu	**Tools® Process Specific® Progressive Die Wizard® Scrap Design**

Bill of Material

What is it?

In the **BOM Record Edit** dialog box, you can now:

- Have as many columns as you need.
- Sort and edit columns easily.
- Enter up to 132 characters in each string.
- Perform actions from the shortcut menu instead of using buttons.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Bill of Material**

Menu **Tools® Process Specific® Progressive Die Wizard® Bill of Material**

Pocket Design — Select pocketing body

What is it?

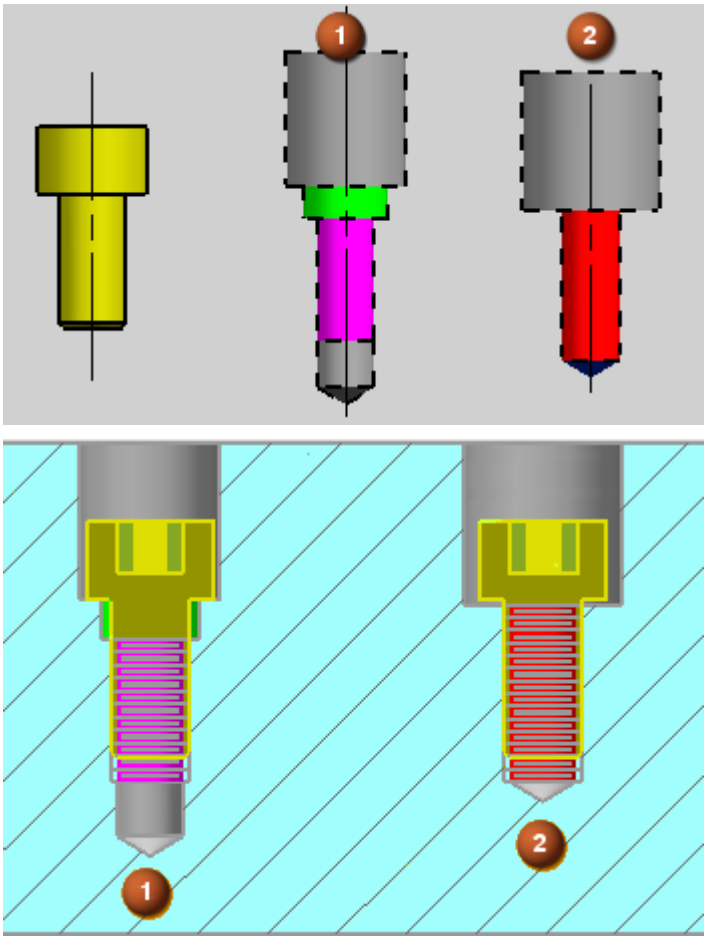
In the **Pocket Design** command, you can now:

- Select a body from a **FALSE** reference set with multiple bodies.
- Show or hide shortcut buttons for the two choices in the **Select Types** list.

The magenta face in body (1) and the red face in body (2) have the **PDW_HOLE_THREAD** attribute set to **1**.

Body (1) is modeled to provide a short clearance distance at the top of the **M8x1.25** threaded hole, represented by the green face. The tap drill diameter in body (1) extends about 10 mm beyond the threaded distance.

Body (2) has a larger counterbore diameter than body (1). The full length of the tap drill diameter of body (2) is threaded.



Why should I use it?

You can model different **FALSE** reference set bodies to provide a choice of pocket shape, counterbores, clearances, and threaded distance.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Pocket Design**
Menu	**Tools® Process Specific® Progressive Die Wizard® Pocket Design**
Location in dialog box	**Tool** group, **Select Types**

Pocket Design — Add material

What is it?

In the **Pocket Design** command, you now have two modes of pocket creation:

- **Subtract Material**
- **Add Material**

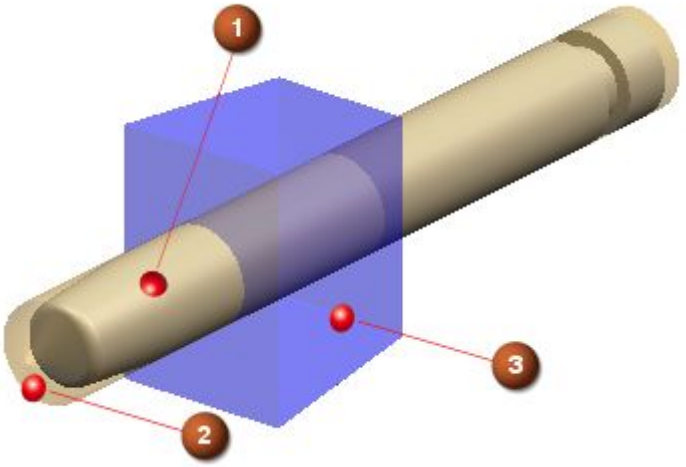
You can display the new **ADD_MATERIAL** reference set to show bodies that represent added material.

The first figure shows a mold lock pin that requires both a subtracted hole shape, and an added tapered pad.

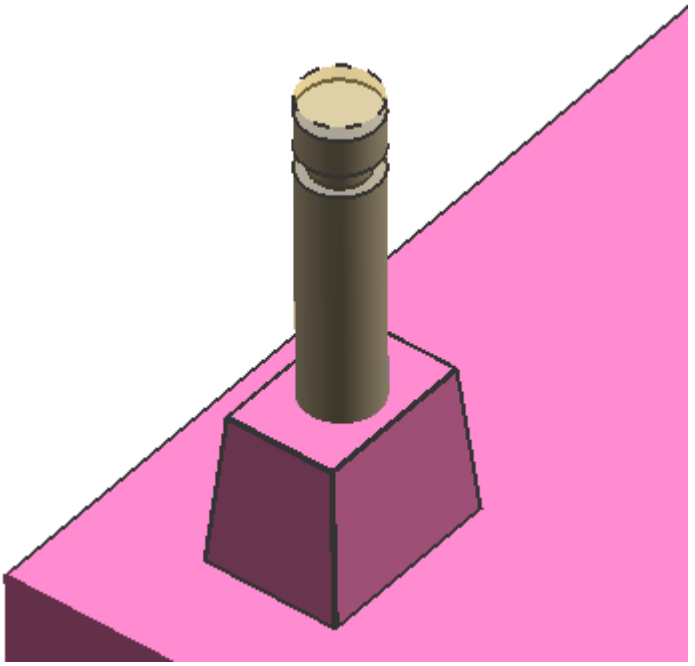
The pin (1) is in the **TRUE** reference set.

The hole body (2) is in the **FALSE** reference set.

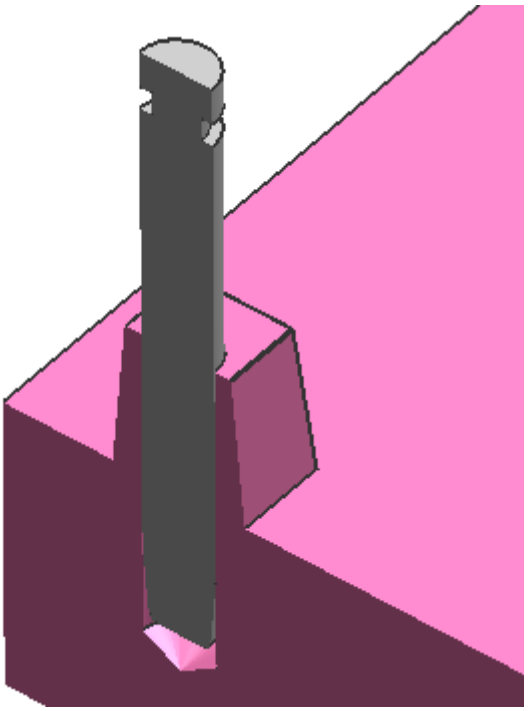
The pad body (3) is in the **ADD_MATERIAL** reference set.



The following figure shows a plate modified by using the **Add Material** pocket mode with the **ADD_MATERIAL** reference set.



The following figure shows the plate after a second pocket operation using the **Subtract Material** mode with the **FALSE** reference set.



Why should I use it?

Use it when you need to add material to a target body to properly represent the contours needed to position a standard part.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Pocket Design**
Menu	**Tools® Process Specific® Progressive Die Wizard® Pocket Design**
Location in dialog box	**Mode® Add Material**, **Reference Sets® ADD_MATERIAL**

New parts created by rename

What is it?

You can use the new default rename method instead of the clone method to create new component parts when you use:

- The **Die Base Management** dialog box.
- Any command that uses the Standard Part Management system.

You can change the default method using the **Part Installation Method** customer defaults option.

Why should I use it?

The rename method is faster than the clone method, and it does not automatically save new parts.

When you close new parts without saving them, you no longer have to remove unwanted cloned files from your project folder.

Where do I find it?

Commands

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard**
Menu	**Tools® Process Specific® Progressive Die Wizard**
Commands supported	**Initialize Project**
	Standard Parts
	Die Base
	Piercing
	Insert Group
Location in dialog box	File parameters and file commands sub window

Part Installation Method customer default

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Progressive Die Wizard®** General page

Stock Size

What is it?

The new **Stock Size** button opens the **Edit Stock Size** dialog box from the **Progressive Die Tools** toolbar. This means that you can now assign a stock size to a part without having to open the **BOM Record Edit** dialog box.

You can still edit existing stock sizes with options in the shortcut menu and the **Settings** group of the **BOM Record Edit** dialog box.

Where do I find it?

Application	Progressive Die Wizard
Toolbar	**Progressive Die Wizard® Progressive Die Tools® Stock Size**
Menu	**Tools® Process Specific® Progressive Die Wizard® Progressive Die Tools® Stock Size**

One-step Formability Analysis

What is it?

See One-step Formability Analysis for more information.

Weld Assistant

BIW Locator

What is it?

The new **BIW Locator** toolbar has new commands that enable you to define datums and measurement point features in the 3D models used in the manufacturing of automotive bodies.

Use the new commands to:

- Create and locate a datum feature, or a measurement feature which can be used as an input for manufacturing tooling and checking systems.
- Import and export datum and measurement features as comma separated value (CSV) files.

Note

Along with the new commands the **Connected Face Finder**, **Weld Advisor**, **Display Weld CSYS in Work Part/ Display Weld CSYS in Work Assembly**, and **Show Through in Work Part/ Show Through in Assembly** commands are also available on the **BIW Locator** toolbar and work the same as the commands available on the **Weld Assistant** toolbar.

Why should I use it?

Use the datum point and datum measurement features you create to define critical areas of a vehicle body that must be measured in order to ensure proper form and fit of body panels.

Where do I find it?

Application	Modeling
Toolbar	**BIW Locator**

For information about the new commands on the **BIW Locator** toolbar, see:

Datum Locator

What is it?

Use the **Datum Locator** command to specify critical areas of the assembly that need to be measured, monitored, and coordinated.

You can:

- Insert a Datum Locator feature on a surface, hole, pin, or slot.
- Use rules, values, and parameters to position the Datum Locator feature.
- Generate the feature on faces or along curves if face geometry is not available.
- Make the feature associative to the creation geometry or freeze it in the coordinate face.
- Add or remove faces, curves, and edges from the Datum Locator feature set. You can do this only when they do not impact the location of the feature and are included in the coordination checks.
- Specify the cross section to be used.
- Enter a name for the feature.

Customer defaults are provided to customize the creation of Datum Locators.

Why should I use it?

The Datum Locator feature is a required input for specifying datum points and contains rules for the tolerance zone, which is used to evaluate coordination.

Where do I find it?

Datum Locator command

Application	Modeling
Toolbar	**Weld Assistant® Datum Locator**
	BIW Locator® Datum Locator
Menu	**Insert® Welding® Datum Locator**

Datum Locator customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Customer Defaults® Weld Assistant® Datums**

Measurement Locator

What is it?

Use the **Measurement Locator** command to define the location of measurement points and assign attributes.

You can:

- Create a Measurement Locator feature on a surface, hole, slot, stud, trim, or hemmed edge.
- Use rules, values, and parameters to position the Measurement Locator feature.

Customer defaults are provided to customize the creation of Measurement Locators.

Why should I use it?

Measurement points are required to specify a particular area of a part that needs to be measured in order to determine whether a part is within the acceptable tolerance.

Where do I find it?

Measurement Locator command

Application	Modeling

Toolbar	**Weld Assistant® Measurement Locator**
	BIW Locator® Measurement Locator
Menu	**Insert® Welding® Measurement Locator**

Measurement Locator customer defaults

Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Customer Defaults® Weld Assistant® Measurement Points**

Import Datums

What is it?

Use the **Import Datums** command to import Weld Assistant point locations from other weld or legacy systems in the form of a comma separated values (CSV) file. This command works the same as the **Import Welds** command available on the **Weld Assistant** toolbar.

Where do I find it?

Application	Modeling
Toolbar	**BIW Locator® Import Datums**

Weld Filter

Type Filter and Detailed Filtering – Weld types options

What is it?

The following **Type Filter** and **Detailed Filtering** options are available on the Selection bar.

Type Filter option	Corresponding Detailed Filtering – Detailed Types option

Welding Objects	**Resistance Spot**
	Arc Spot
	Mechanical Clinch
	Dollop
	Nut
	Stud
	CUSTOM WELD 1
	CUSTOM WELD 2
	CUSTOM WELD 3
	CUSTOM WELD 4
	CUSTOM WELD 5
	Groove
	Fillet
Body in White Datum	**Surface**
	Pin
	Datum Custom 1
	Datum Custom 2
	Datum Custom 3
Measurement Objects	**Surface**
	Hole
	Stud
	Slot
	Trim
	Hemmed Edge
	Measurement Custom 1

Measurement Custom 2

Measurement Custom 3

Where do I find it?

Type Filter

Application	Gateway
Toolbar	Selection bar
Main Window	**Type Filter**

Detailed Filtering

Application	Gateway
Toolbar	Selection bar® **Detailed Filtering**
Location in dialog box	**Detailed Filtering**® Types page **Detailed Filtering**® Detailed Types page

Weld Attribute Filter

What is it?

Use the **Weld Attributes** command to specify qualified weld attributes as filters on the Selection bar.

You can:

- Specify the weld attributes such as **Welding Objects**, **Body in White Datum**, or **Measurement Objects** to narrow down the selection range.
- Inherit existing attributes from current objects, as identified by the **Type Filter** on the Selection bar, and add them to the global filtering options.

  Note

  The **Weld Attributes** command is not available on the Selection bar by default. To use this command, you need to customize the Selection bar.

Where do I find it?

Application	Gateway
Prerequisite	**Type Filter** must be set to **Welding Objects**, **Body in White Datum**, or **Measurement Objects**.
Main window	Selection bar® **Weld Attributes**

Weld Point

What is it?

The **Weld Point** command has new **Spacing Method** options that enable you to create weld points using a combination of guide curve and specific coordinate value. You can create single or multiple points.

Weld point spacing is:

- Calculated along the length of the guide curve when you select the **Arc Length** option.
- Determined by parallel planes along the X, Y or Z axes when you select the **Parallel X Plane**, **Parallel Y Plane** or **Parallel Z Plane** options.

Why should I use it?

Use these options to easily locate weld points at a precise coordinate value while maintaining adherence to an offset guide edge and reference sheet. For example, in automotive design, you can use the vehicle body coordinates to define the placement of spot welds.

Where do I find it?

Application	Modeling
Toolbar	**Weld Assistant**® **Weld Point**
Menu	**Insert**® **Welding**® **Weld Point**
Location in dialog box	**Positions Options** group® Model page® **Spacing Method** list

Auto Point

What is it?

Auto Point is a new command that provides a fast and simple way to determine the overlapping regions of an assembly, and generate weld points with minimal user input.

Note

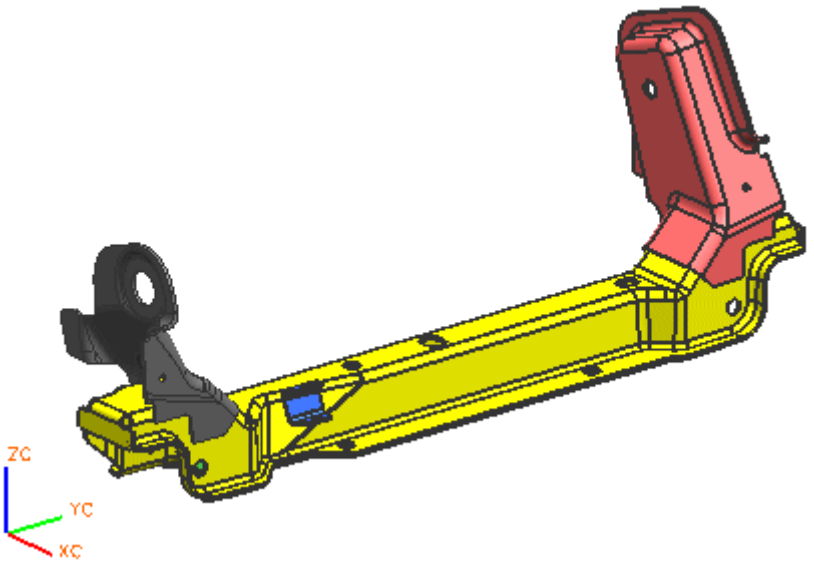
You can edit the input used to generate each string of weld points using the **Weld Point** command.

Use the **Auto Point** command to:

- Perform initial structural integrity studies early in the design cycle by creating weld points quickly where components overlap.

- Quickly locate where two, three or four panels are joined.
- Assigns the weld IDs and connected part information early in the weld life cycle.
- Find overlap regions and then use the **Weld Point** command to adjust point locations.
- Exclude existing sub-assemblies from auto spot weld generation.

The following animation shows weld points created automatically using only the assembly components as input.



Where do I find it?

Application	Modeling
Toolbar	**Weld Assistant® Auto Point**
Menu	**Insert® Welding® Auto Point**

Export Welds

What is it?

Export Welds is a new command that enables you to write spot weld information to a comma separated value (CSV) file.

With **Export Welds** you can:

- Specify the spot welds and the attributes that should be output.
- Configure the order in which the attributes are written.

- Save the attributes and order as a template to be used for subsequent outputs.
- Export spot welds from one NX assembly and import them in another NX assembly.

Why should I use it?

Spot weld data written to a neutral format is more easily used by another application or database. Export templates provide a consistent output format and different users can share them.

Where do I find it?

Application	Modeling
Toolbar	**Weld Assistant**® **Export Welds**
Menu	**Insert**® **Welding**® **Export Welds**

Group ID colors

What is it?

Group ID Color is a new option in the **Weld Preferences** dialog box that enables you to assign colors to spot welds based on Group ID assignment. You can:

- Predefine these colors in the **Customer Defaults** dialog box.
- Use the **Group ID Color** option in the **Weld Preferences** dialog box to control color assignment.

Why should I use it?

Use Group ID colors to visually distinguish spot weld groups in your model.

Where do I find it?

Group ID Color option

Application	Modeling
Menu	**Preferences**® **Welding**
Location in dialog box	**Weld Preferences**® Weld tab® **Common** group® **Group ID Color**

Group ID Color customer defaults

Menu	**File**® **Utilities**® **Customer Defaults**

Location in dialog box	**Customer Defaults® Weld Assistant® Common** ® Group ID Color page

Chapter

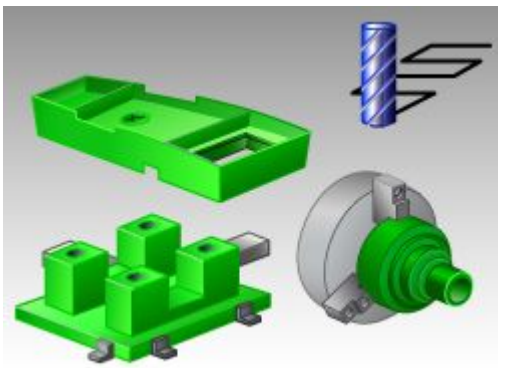
14 Manufacturing

General

Manufacturing setups available with File→New

What is it?

You can now create a manufacturing setup assembly when you select a default Manufacturing template in the **New** dialog box. The default templates help you create CAM Express setups, a general setup, or a blank.



General Setup

See also:

- *Creating a new part file* in the Gateway to NX online Help.

Why should I use it?

This is the recommended way to create a CAM setup using an assembly.

Where do I find it?

Application	Manufacturing
Prerequisite	You must reference an NX model that contains the part geometry.
Menu	**File® New**
Location in dialog box	Manufacturing page

Operation Navigator

Operation Navigator functions

What is it?

The **Operation Navigator** has the following enhancements:

- A selected operation remains selected when you switch views.
- Tool paths and geometry are displayed in the graphics area when you select them in the **Operation Navigator** so that you can quickly browse and see what is defined and the areas machined. This option can be turned off in **Manufacturing Preferences**.
- A turning operation In Process Workpiece (IPW) can be displayed.
- Operations can now be cut or copied and pasted into another part. You can use this functionality instead of creating an operation template.

  Note

  You can only copy and paste between parts with the same units (inch or metric).

- Columns for Start Events and End Events let you find the operations with user defined events without editing each operation.
- Objects that are in the **Unused Items** folder of any view are displayed with a different text color in all views.

See also:

- Operation Navigator customer defaults

Where do I find it?

Application	Manufacturing
Resource bar	**Operation Navigator**

Operation Navigator status indicators

What is it?

New tool path status indicators include:

- **Empty Tool Path**

  The path has been generated, but does not contain valid motion. Some examples are a **Flowcut** operation that finds no valleys, or a **Cavity Mill**

operation without an IPW to cut. Operations, like **Machine Control**, that are not designed to contain motion will not display this status.

- **Locked**

  The tool path is protected from overwriting.

  Use this if you have edited a tool path, or used a function like flowcut manual cut order and do not want to overwrite the tool path accidentally.

  The **Locked** status is also displayed when you open a file saved in a previous release if the**Customer Defaults** option **Lock Tool Paths during version upgrade** is selected.

A warning message is displayed if you attempt to generate an operation with a status of **Imported**, **Edited**, or **Locked**.

A new operation status has been added:

- **Approved**

  The tool path is acceptable in its current state.

  A geometry or tool parameter change will set operations to **Regenerate** status. If you know that the tool path is still up to date, you can override the **Regenerate** status by changing it to **Approved**.

Where do I find it?

Application	Manufacturing
Prerequisite	Operations with tool path generated.
Shortcut menu	**Operation Navigator**® right–click the operation® **Tool Path**® **Lock** and **Object**® **Approved**

Operation Navigator customer defaults

What is it?

New customer defaults are added that control functions in the **Operation Navigator**.

- **Select operations in all views** keeps a selected operation selected even when you change views.

- **Display selected objects** highlights the contents in the graphics window.

  When examining objects in the **Operation Navigator**, displaying the selected objects makes it easier to identify what each object contains. You may want to turn this option off for large tool paths.

- **Display Turning IPW of selected objects** displays a turning operation In Process Workpiece.

- **Show tool descriptions in Create Operation** displays a brief description next to each tool name in an operation's **Tool** list. For example, *(Drilling Tool)* or *(Turning Tool-Standard)*.

  Sometimes it is confusing to select the correct tool from a drop down list, especially with library tools. Showing the tool description makes it easier to identify the desired tool.

- **Lock Tool Paths during version upgrade** offers some protection from accidentally regenerating old tool paths when you open a file saved in a previous release.

See also:

- Operation Navigator

Where do I find it?

Application	Manufacturing
Prerequisite	To display the turning In Process Workpiece (IPW), you must have a turning operation with an IPW generated.
Menu	**File**® **Utilities**® **Customer Defaults** **Customer Defaults** dialog box® **Manufacturing**® **User Interface**® Operation Navigator page
Location in dialog box	**Customer Defaults** dialog box® **Manufacturing**® **User Interface**® Dialogs page

Note

The **Display selected objects** and **Display Turning IPW of selected objects** options can be turned off in **Manufacturing Preferences**.

Template changes

What is it?

There are several template changes that affect dialog boxes.

- There are two new templates: **probing** and **machining_knowledge**.
- Legacy lathe was removed as a template type.
- Zig Zag surface was removed from the **mill_multi-axis** operation templates.
- **Tool Axis** is in the main dialog box of Cavity Milling, Plunge Milling, and Zlevel operations.
- Motion output is available in the **Machine Control** group of most operations.

- **Contour Profile** operations inherit part stock.

If the current templates do not match your workflow, see *Customize Dialog Options* for information on how to add or remove dialog box options.

Corner Control

What is it?

Corner options previously in the **Corner Control** or drive method dialog boxes are now available on the Corners page of the **Cutting Parameters** dialog box.

Where do I find it?

Application	Manufacturing
Location in dialog box	**Cutting Parameters** dialog box® **Corners** tab

Cut Patterns

What is it?

Milling operations now use consistent terminology and presentation order for cut patterns. The **Pattern** and **Cut Type** options for Surface Contouring have been replaced with the single option **Cut Pattern**. For example:

Previous **Pattern** and **Cut Type** options	New **Cut Pattern** option
Parallel Lines, **Zig-Zag**	**Zig Zag**
Radial Lines, **Zig**	**Radial Zig**

Why should I use it?

Consistent terminology makes it easier to use the software.

Where do I find it?

Application	Manufacturing Planar operations: Main dialog box® **Path Settings** group
Location in dialog box	Surface contouring operations: Drive Method dialog box® **Drive Settings** group

CAM roles

What is it?

CAM configuration is now saved in the role. Changing your role changes the CAM configuration. You can turn off the **Change Configuration Based on Role** customer default if you use another method to set the CAM configuration.

Why should I use it?

The configuration file controls which operation types and subtypes are available. You do not need to exit Manufacturing to change the configuration file.

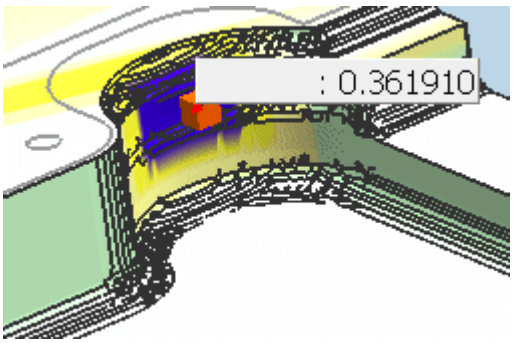
Where do I find it?

Application	Manufacturing
Menu	**File**® **Utilities**® **Customer Defaults**
Location in dialog box	**Customer Defaults** dialog box® **Manufacturing**® **User Interface**® Roles page
Resource Bar	**Roles**

IPW color plots

What is it?

The **Show Thickness by Color** option for 2D and 3D verification creates a color–coded plot that shows the minimum distance between the In Process Workpiece (IPW) and the design part. Select points in the graphics window to display the amount of material left on the part.



Why should I use it?

Use **Show Thickness by Color** to optimize machining procedures by identifying areas on the part with excess material remaining.

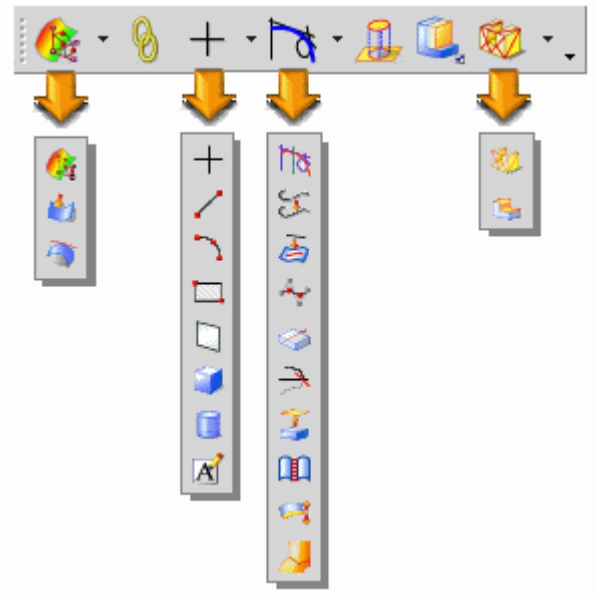
Where do I find it?

Application	Manufacturing
Prerequisite	A milling operation with tool path generated and an available 3D In Process Workpiece
Resource bar	**Operation Navigator**® right-click the operation® **Tool Path**® **Verify** **Operation Navigator**® right-click the operation® **Workpiece**® **Show Thickness by Color** **Operation Navigator**® right-click the operation® **Tool Path**® **Simulate** **Tool Path Visualization** dialog box® **3D Dynamic** and **2D Dynamic** tabs
Location in dialog box	**Simulation Control Panel** dialog box® **Animation Settings** group

CAM Geometry toolbar

What is it?

CAD tools are now available in one convenient location for CAM users. The **CAM Geometry** toolbar includes the commands previously available with **Prepare Geometry**.



Why should I use it?

CAD tools let you edit the part model for machining without leaving the Manufacturing application.

Where do I find it?

Application	Manufacturing
Toolbar	**Geometry**

Command Finder for Manufacturing

What is it?

Command Finder now includes descriptions of all operations, tools, geometry groups, methods and programs.

Why should I use it?

You can locate a command in NX by entering either the full command name or a part of the name in **Command Finder**.

If you do not know the command name, you can find a command by searching on a term related to that command. For example, enter **IPW** to see IPW-related commands, and operations that use an In Process Workpiece.

Where do I find it?

Toolbar	**Standard® Command Finder**
Menu	**Help® Command Finder**

Journaling and API

What is it?

Journaling and API coverage for operations includes:

- Operation parameters for operations with the block based dialog boxes
- Non cutting moves
- Cutting parameters
- Surface Contouring drive parameters
- Feeds and Speeds

Journaling and API coverage for the **Operation Navigator** includes:

- Transform operations
- Copy/Paste operations between parts
- Approve operations

- Lock tool paths
- Feed rates

There are several *.net* examples that you can use to write your own programs.

Where do I find it?

Application	Manufacturing
File location	*${UGII_BASE_DIR}\ugopen\SampleNXOpenApplications\.Net\C*

CAM Express

CAM Express home page

What is it?

From the CAM Express home page, you can now open Solid Works files and create a customized link that opens a part.

The parts must be in the *mach\tutorials\parts* directory. The naming convention is:

CAM_TU_<part_name>.pax

CAM_TU_ASSEM_<part_name>.pax

Where do I find it?

Application	Manufacturing
Resource Bar	**Roles** ® Industry Specific folder® CAM Express

CAM Express tutorials

What is it?

CAM Express has four new tutorials.

High Speed Machining

This tutorial shows you how to:

- Apply the latest best practices.
- Use the Machining Data library.

Aerospace Machining

This tutorial shows you how to create multi-axis tool paths on an aerospace part.

Mold Rework

This tutorial shows you how to:

- Modify an existing program.
- Apply localized machining.

Postprocessor Download and Edit

This tutorial shows you how to:

- Install a postprocessor.
- Edit the postprocessor with Post Builder.

Where do I find it?

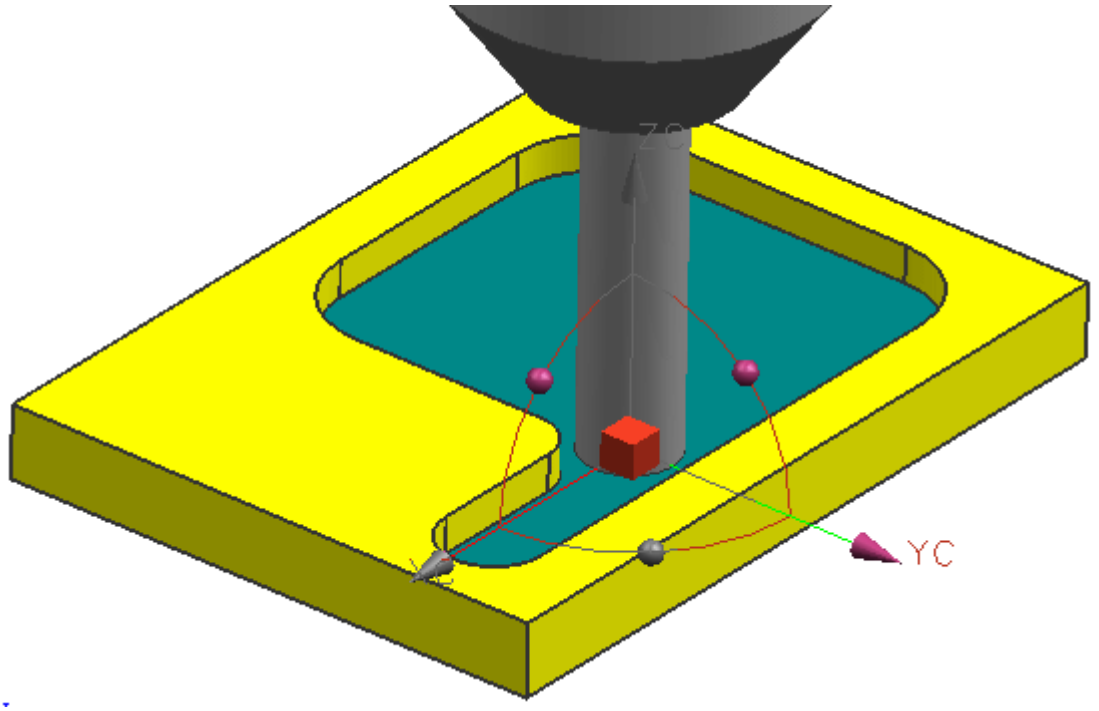
Application	Manufacturing
Resource bar	**CAM Express** home page

Tools

Tool display

What is it?

Tools are displayed as a solid with handles that allow you to drag and rotate the tool around the part.



Why should I use it?

Use this enhancement to verify the tool dimensions against your part as you enter them.

Where do I find it?

Application	Manufacturing
Location in tool dialog box	**Preview** group® **Display**
Location in operation dialog box	**Tool** group

Pockets

What is it?

You can now:

- Specify an adjust register and cutcom register in the **Pocket** dialog box and the tool will inherit these values. By default, they both will match the tool number.
- Modify the list of holding systems on the pocket.

Pockets now inherit adjust register and cutcom register from the machine model.

Why should I use it?

Use this enhancement to specify an adjust or cutcom register that is different than the tool number. Use the holding systems when you want to restrict tool loading. For example, you may have some positions on a turret that accept one holding system, and other positions that accept a different holding system.

Where do I find it?

Application	Manufacturing
Menu	**Insert**® **Tool**
Toolbar	**Manufacturing Create**® **Create Tool** .
Location in dialog box	**Tool Subtype** group® **MCT_POCKET**

Turning tools

What is it?

Enhancements include:

- The **User Defined** options for **Insert Shape** and **Hand** let you parametrically define insert and holder combinations that were previously not available.

- Support for solid tools.

  You can define a 2D shape within a solid library tool that represents the lathe tool holder and is used for collision checking against the 2D geometry of the lathe workpiece.

- Round-shanked parametric turning tools are displayed with cylinders instead of flat segments.

Where do I find it?

Application	Manufacturing
Prerequisite	Turning operation
Location in dialog box	**Tool** group

Solid tools for Turning

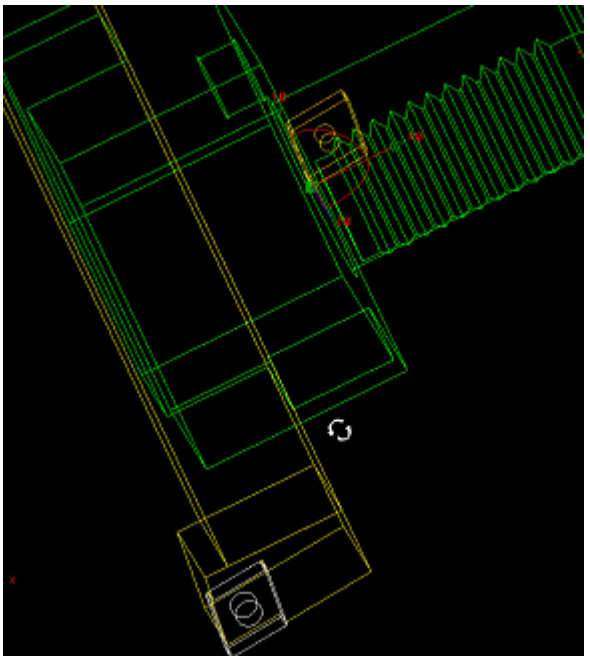
What is it?

Turning now supports solid tools. You can:

- Open the solid model of a tool and use it to specify the tool's parameters.
- Create multiple tracking points in the solid model.
- View the tool's coordinate system displayed at the currently selected tracking point.
- Add the tool's solid model to the tool library.

There are two new parameters with the **Export Tool Part File** option:

- **Tool Mount Junction** locates the tool within the machine tool assembly.
- **Tool Tip Junction** defines the first corner radius (R1) of the cutter. Its X/Y plane defines the work plane of the tool and the orientation angle and tracking point P–Number refer to it.



Why should I use it?

With these enhancements you can:

- Use the tool assembly display to more easily define a tool's tracking points.
- Apply the turning operation's tracking change functionality benefits to solid turning tools.

- Use a solid tool's model for the graphics display in Turning and ISV.

Where do I find it?

Application	Manufacturing
Prerequisite	Solid model of a turning tool
Menu	**Insert® Tool**
Location in dialog box	Turning tool dialog box® **Library** group® **Export Tool Part File**, **Mounting Junction**, **R1 Tool Tip Junction**
File location	Solid tool model: *MACH/resource/library/tool/graphics* Parameter files *tool_database.dat* and *trackpoint_database.dat* : *MACH/resource/library/tool/english* or */metric*

Milling and Drilling tools available in Turning

What is it?

You can now:

- Use the following parametric milling tools for roughing, finishing and teach mode operations.
  - 5-Parameter
  - Face mill
  - Ball mill
  - T-Cutter

  In the **Machine Tool** view, drag the operation under the tool and then edit the operation to re–orient the tool axis.
- Use the available parametric drilling tools for centerline drilling, roughing, finishing, and teach mode operations.

The IPW updates properly for the tool.

Why should I use it?

With these enhancements you can:

- Use more tool options to match part shapes.

  For example, you can use an end mill to counter bore.
- Reduce tool changes.
- Reduce chip length for some operations.

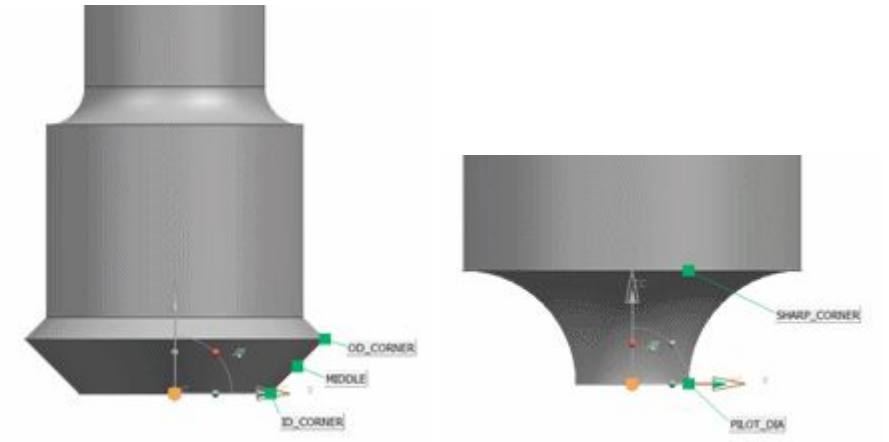
Where do I find it?

Application	Manufacturing
Prerequisite	Turning or centerline drilling operation
Location in dialog box	**Tool** group

Sample user defined milling tools in the tool library

What is it?

The tool library now includes sample user defined tools for corner rounding and chamfering that include meaningful tracking points.



Why should I use it?

Use these with planar profile operations to round or chamfer edges.

Where do I find it?

Application	Manufacturing
Toolbar	**Insert**® **Create Tool**
Menu	**Insert**® **Tool**
Location in dialog box	**Create Tool** dialog box® **Retrieve Tool from Library**® **Library Class Selection** dialog box® **Milling**® **Mill Form Tool**

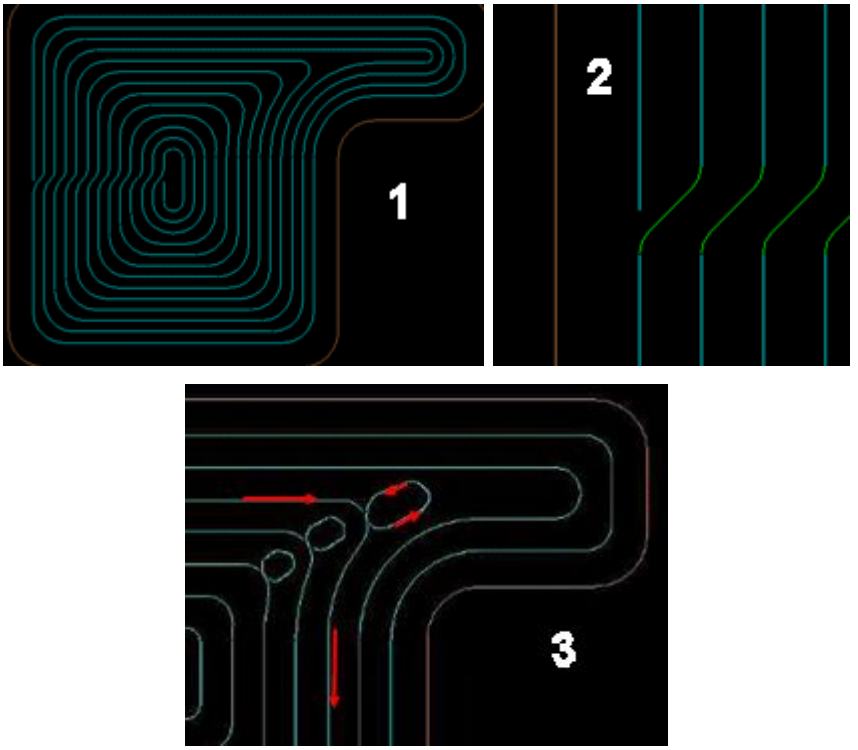
Milling

Tool path smoothing

What is it?

Enhancements include adding:

- Smooth corners.
- Smooth stepovers so that the tool does not stop in the direction of cut.
- Smooth loops when uncut material is detected.



Corners (1), stepovers (2), loops for uncut material (3)

Why should I use it?

Smoother tool paths are more efficient for high speed machining.

Where do I find it?

Application	Manufacturing
Prerequisite	Milling operation
Location in dialog box	**Cutting Parameters** dialog box® **Corners** tab

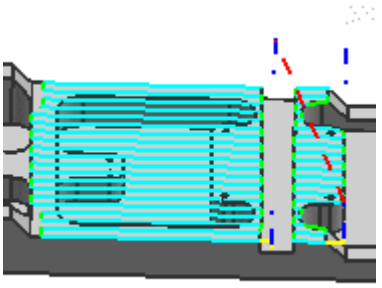
Face Milling

Merge Distance

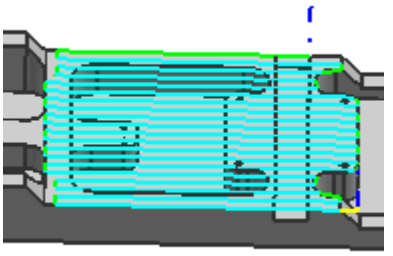
What is it?

The **Merge Distance** option lets you machine multiple cut area faces or blank boundaries as a single cut region if they are:

- On the same level.
- Within the specified distance of each other.



Small **Merge Distance**



Increased **Merge Distance**

Why should I use it?

Use **Merge Distance** to minimize lifts within the tool path.

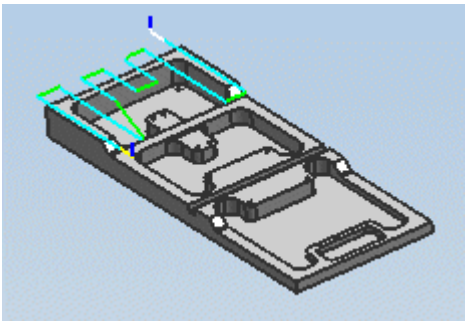
Where do I find it?

Application	Manufacturing
Prerequisite	Face Milling operation
Location in dialog box	**Cutting Parameters** dialog box® Strategy page® **Blank** group

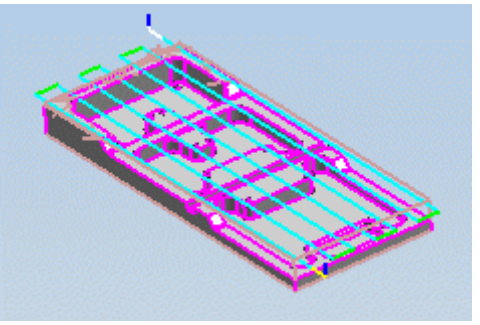
Extend blank to the part outline

What is it?

The **Extend to Part Outline** option extends the blank definition normal to the tool axis to the outline of the part geometry.



Tool path from selected face



Tool path extended to outline of part

Why should I use it?

Use this option to quickly and easily remove all the material that lies above a particular level of the part. You can face off the top of a part by extending the cuts of the highest face up to the edges of the blank.

Where do I find it?

Application	Manufacturing
Prerequisite	Face Milling operation
Location in dialog box	**Cutting Parameters** dialog box® Strategy page® **Blank** group

Machining across voids

What is it?

The option to machine across voids is now available when using a cut area.

The **Cut** option for **Motion Type** cuts across all internal voids. Previously this was controlled by the **Ignore Holes** option when specifying blank boundaries.

Why should I use it?

To simplify cutting over holes.

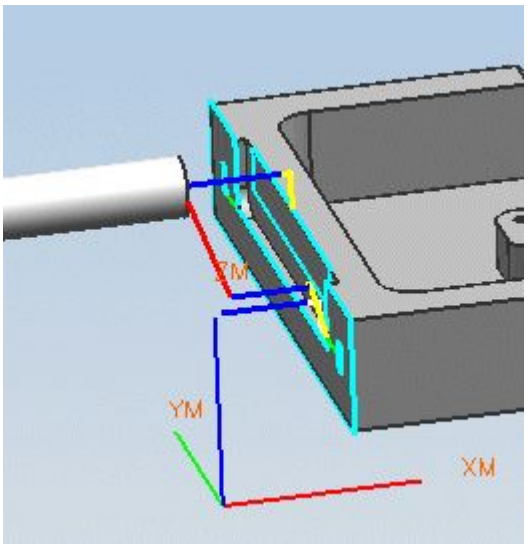
Where do I find it?

Application	Manufacturing
Prerequisite	Face Milling operation
Location in dialog box	**Cutting Parameters** dialog box® Connections page® **Across Voids** group

Normal to First Face

What is it?

The **Normal to First Face** option makes the tool axis normal to the first selected cut area or blank boundary face.



Tool axis Normal to First Face

Why should I use it?

With this option, the software automatically uses the first face selected to set the tool axis.

Where do I find it?

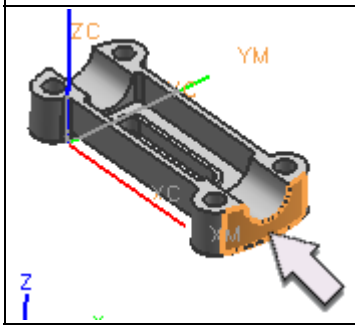
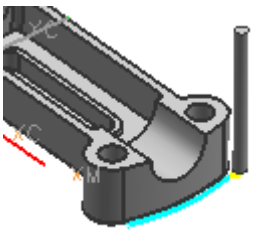
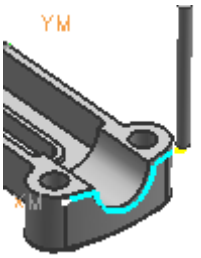
Application	Manufacturing
Prerequisite	Face Milling operation
Location in dialog box	**Tool Axis** group

2D/3D Profiling

Solid Profile 3D operation

What is it?

The new **Solid Profile 3D** operation lets you machine either the top or bottom edge of a vertical wall without selecting the individual edge chains.

Face selected	Bottom edge profiled	Top edge profiled
		

Where do I find it?

Application	Manufacturing
Toolbar	**Insert**® **Create Operation**
Menu	**Insert**® **Operation**
Location in dialog box	**Type** group® **mill_contour** **Operation Subtype** group® **SOLID_PROFILE_3D**

Collision Check

What is it?

2D Profile, **Profile 3D**, and **Solid Profile 3D** operations include collision checking.

Why should I use it?

This provides an easy and automatic way to avoid gouges.

Where do I find it?

Application	Manufacturing
Prerequisite	**2D Profile**, **Profile 3D**, or **Solid Profile 3D** operation
Location in dialog box	Main operation dialog box: **Cutting** **Parameters**® **Containment** tab

Mixed cut direction

What is it?

You can now use the **Mixed** option for bi-directional cutting in 2D and 3D profile operations.

Why should I use it?

When climb or conventional cut direction is not essential, the mixed cut direction eliminates unnecessary transfer and rapid moves. This is useful for slotting operations.

Where do I find it?

Application	Manufacturing
Prerequisite	**2D Profile**, **Profile 3D**, or **Solid Profile 3D** operation
Location in dialog box	Main operation dialog box® **Cutting Parameters**® **Strategy** tab

Multiple Passes for Profile 3D and Solid Profile 3D operations

What is it?

Multiple depths and multiple side-passes are now possible in a single **Profile 3D** or **Solid Profile 3D** operation.

Where do I find it?

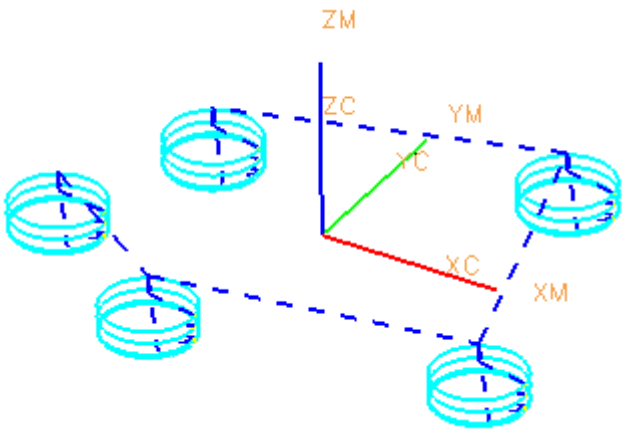
Application	Manufacturing
Prerequisite	**Profile 3D** or **Solid Profile 3D** operation
Location in dialog box	Main operation dialog box: **Cutting Parameters**® **Multiple Passes** tab

Thread Milling

What is it?

Thread milling can now be done manually without thread data in the feature or a data file. Manual thread milling requires placing the thread milling operation in a DRILL_GEOM group. When you do this:

- Locations are from the **DRILL_GEOM** group.
- Pitch is determined by the tool selected.
- User defined parameters do not depend on thread tables or features.
- A new option lets you start at the center.
- Parameters are preserved when you create a template, and when you copy and paste.



Why should I use it?

Use this when you want to thread mill, with known parameters, at any location. Once you establish the tooling and parameters for a thread mill operation, save a template to use for other identical holes.

Where do I find it?

Application	Manufacturing
Menu	**Insert**® **Create** **Operation**® **Type** group® **drill**® **Operation Subtype** group® **Thread Milling**
Location in dialog box	**Thread Milling** dialog box® **Settings** group® **User Parameters**

Fixed and variable surface contouring

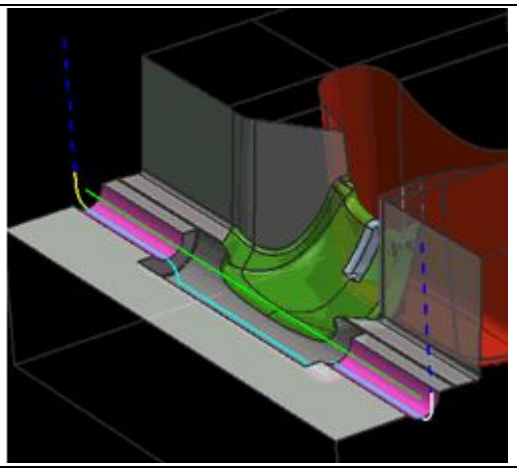
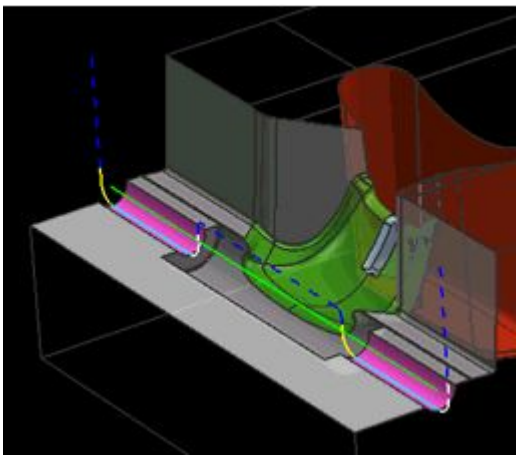
Cut Area support

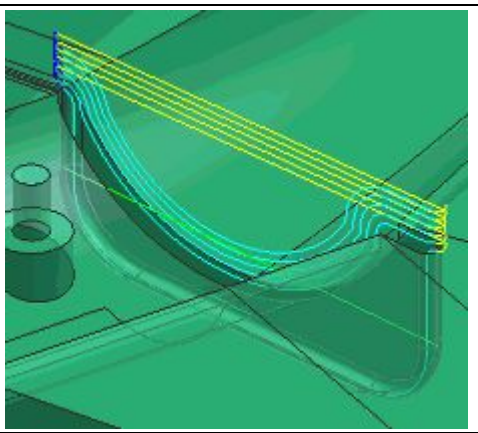
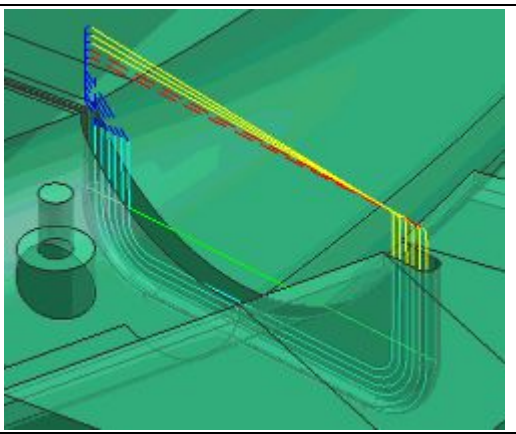
What is it?

All **Fixed Contour** and **Variable Contour** operations now support cut area selection, and offsets for multiple depth cuts are limited to the cut area. The additional drive methods that support cut area selection include:

- **Curve/Point**
- **Boundary**
- **Surface Area**
- **Spiral**
- **Tool Path**

- **Radial Cut**
- **Text**

	
No cut area containment	With cut area containment

	
Multi-depth offsets before	Multi-depth offsets now

Why should I use it?

Selecting a cut area is the easiest way to limit machining to a group of faces. The cut area also lets you easily create uniform multiple depth cuts for features such as slots.

Where do I find it?

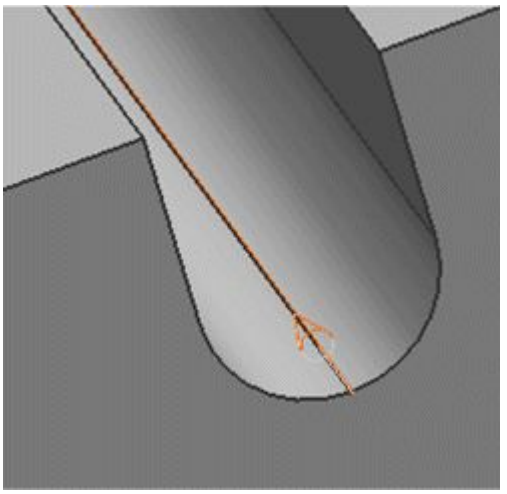
Application	Manufacturing
Prerequisite	**Fixed Contour** or **Variable Contour** surfacing operation.
Location in dialog box	**Drive Method** group **Geometry** group

Curve/Point drive method

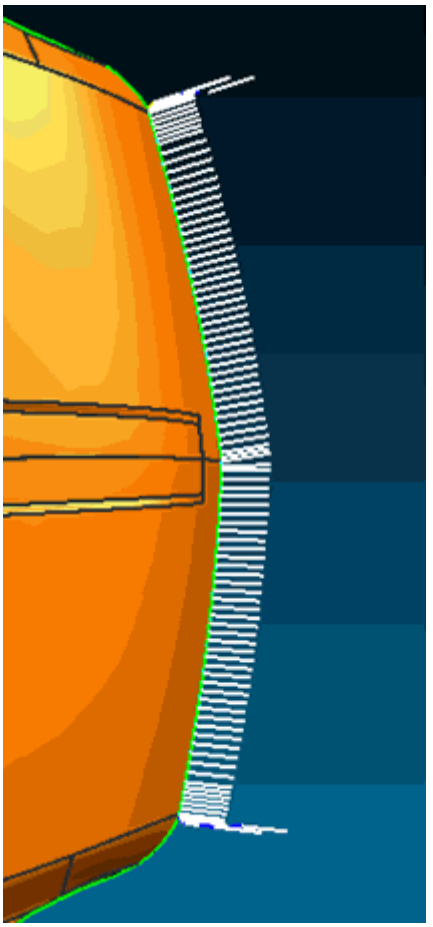
What is it?

Enhancements to the **Curve/Point** drive method for **Fixed Contour** and **Variable Contour** operations include:

- The dialog box is completely redesigned. There is a standard selection list to manage selection sets. **Non Cutting Moves** are used between sets on the list, and replace the local lift at end option.
- **Specify Drive Geometry** now lets you select edges from the part geometry.
- You can use **Normal to Part** for **Tool Axis** when the **Projection Vector** is along the tool axis.



Face edge used for drive geometry



Tool Axis = Normal to Part, Projection Vector is along the tool axis.

Why should I use it?

With these enhancements:

- You can select edges on the part to avoid creating separate wireframe geometry.

- With the new **Projection Vector** and **Tool Axis** combination, you no longer have to manually specify a projection vector. When drive curves are either on or very close to part surfaces, this combination automatically snaps the drive geometry onto part surfaces through the shortest distance and uses part normal as a tool axis.

Where do I find it?

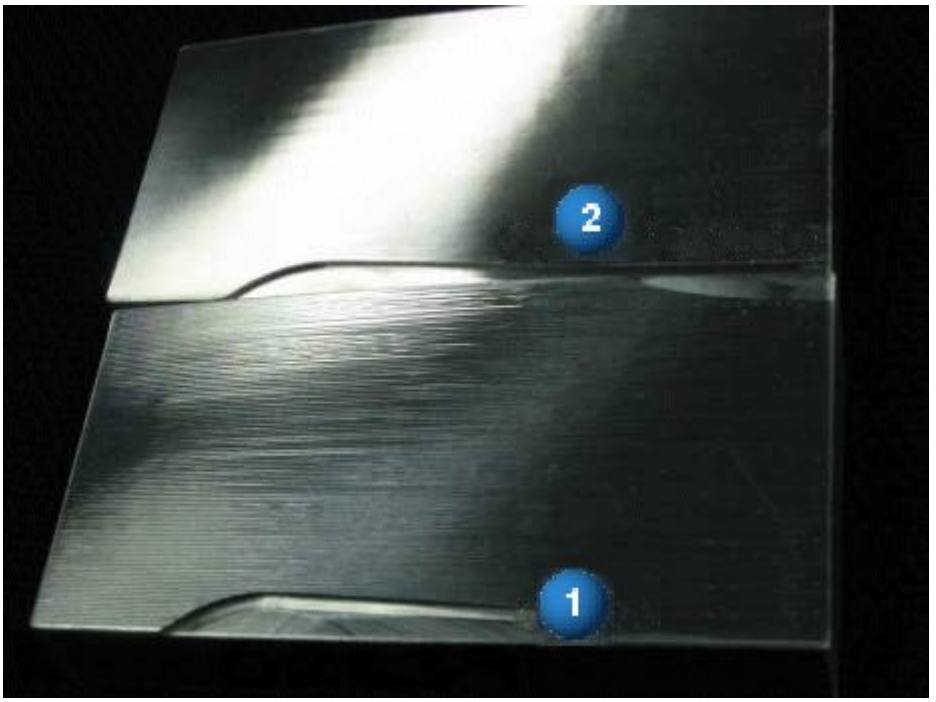
Application	Manufacturing
Prerequisite for projection vector combination	**Variable Contour** surfacing operation.

Location in dialog box | **Projection Vector** and **Tool Axis** group

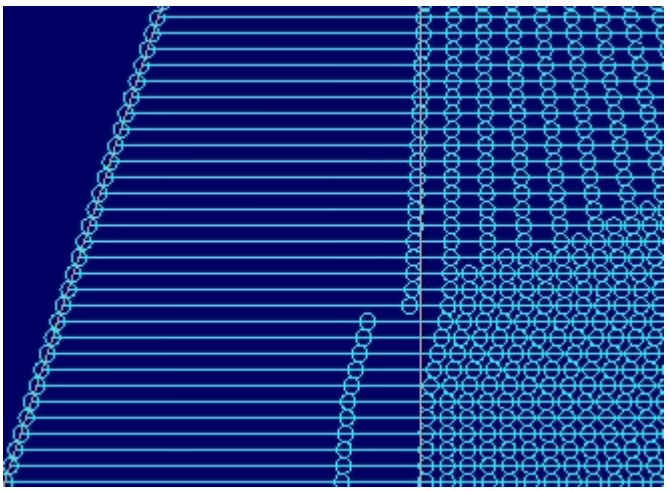
Area Milling point distribution

What is it?

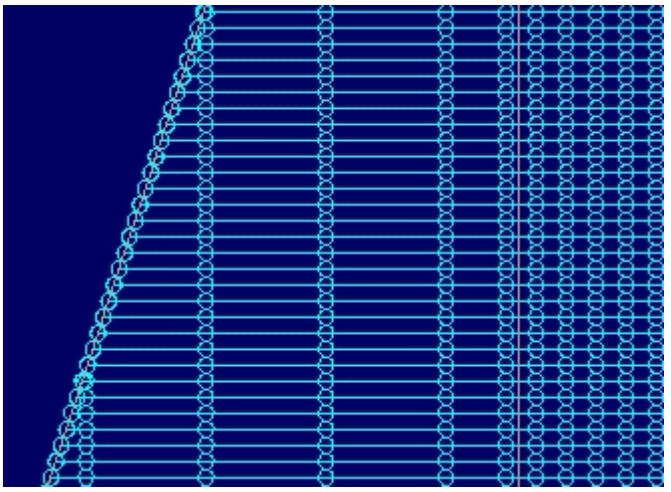
Area Milling operations output a more uniform grid of points on the faces to machine.



Surface finish with old (1) and new (2) point distribution



Old point distribution



New point distribution

Why should I use it?

Use the new point distribution for a finer surface finish on your machined part. You will see the most difference in nearly flat surfaces with slight curvature.

Note

There is a trade-off between tool path size and surface finish. If tool path size is of more importance than surface finish, you can use the NX 5 behavior. For details, see the Manufacturing release notes.

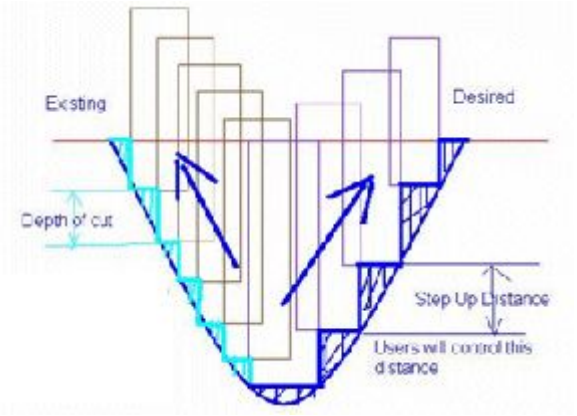
Where do I find it?

Application	Manufacturing
Prerequisite	Area Milling or **Streamline** operations — this is the default behavior The new point distribution is not available for the **On-part** step over method.

Plunge Milling

What is it?

The **Step Up** option lets you control the distance between two or more cut levels.



Why should I use it?

Use **Step up** for improved material control on steep or shallow walls.

Where do I find it?

Application	Manufacturing
Prerequisite	Plunge Milling operation
Location in dialog box	**Plunge Milling** dialog box® **Path Settings** group

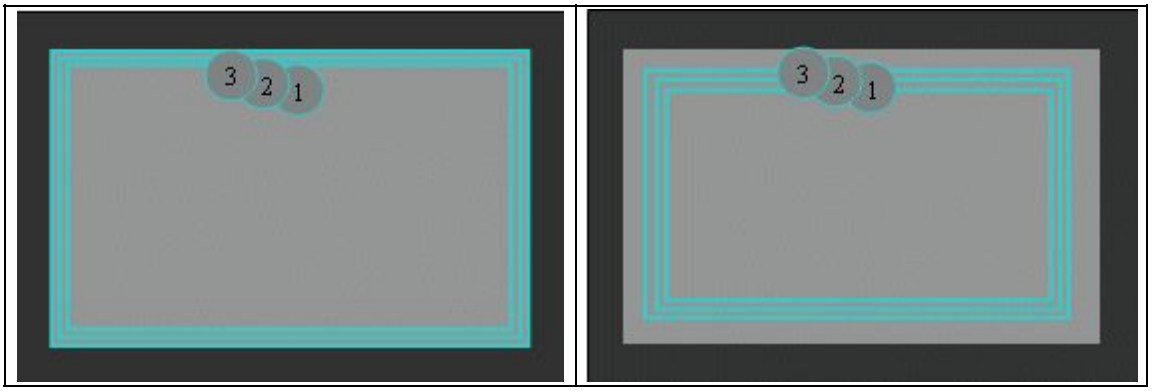
Finish passes and cutter compensation

What is it?

Enhancements to planar machining operations with **Cutter Compensation** let you:

- Specify one or multiple finish passes.
- Specify one of the following for **Finish Passes**:
  - No cutter compensation output
  - Centerline cutter compensation on all passes
  - Contact contour output on all the passes

Contact contour cutter compensation	Centerline cutter compensation



Three consecutive finish passes in a closed pocket

Why should I use it?

Use these enhancements to provide additional passes on the part material for an improved finish.

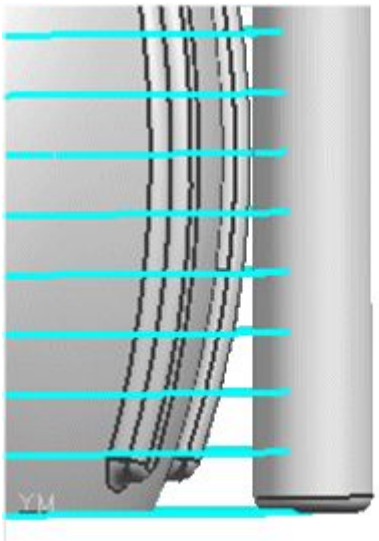
Where do I find it?

Application	Manufacturing
Prerequisite	A fixed axis planar operation
Location in dialog box	**Non Cutting Moves** dialog box® More page® **Cutter Compensation** group

Zlevel

What is it?

The **Continue Cutting Below Tool Contact** option for Zlevel operations lets you cut additional passes on the part.



Where do I find it?

Application	Manufacturing
Prerequisite	Zlevel operation
Location in dialog box	**Cutting Parameters** dialog box® Strategy page® **Extend Path** group

Turning

New look for Turning

What is it?

The Turning dialog boxes have been updated. The layout and terminology for **Cutting Parameters** and **Non-Cutting Moves** are now consistent with Milling.

Why should I use it?

Consistent presentation makes it easier to learn and navigate the software.

Multi-function machines

What is it?

Enhancements for multi–function machines let you:

- Control where retrieved tools are mounted.
- View simulations using the latest revision of a machine.
- View the Operation Navigator and the Machine Tool Configurator (MTC) with their machine tool kinematics and the user interface synchronized.
- Access time calculations by operation, tool, sync regions, etc.
- Use the channel information and flexibility enhancements.
- Access information included in each sync mark for postprocessing.
- Use multiple tracking points.

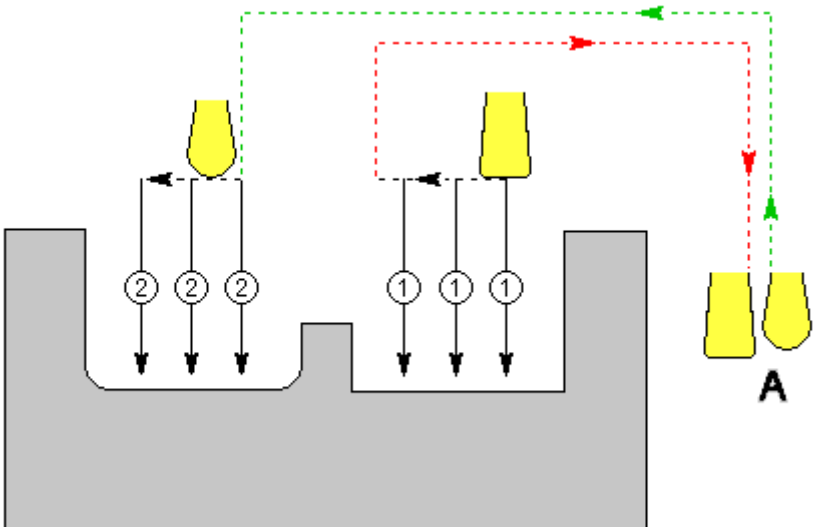
Where do I find it?

Application	Manufacturing
Prerequisite	Turning operation

Approach and Departure

What is it?

You can now apply the Approach or Departure Paths to a tool change (A), or to a special Local Return.



Why should I use it?

Use the **Points Only After Tool Change** and **Points Only Before Tool Change** options to apply an avoidance path to or from the tool change position only if a previous operation uses a different tool.

Where do I find it?

Application	Manufacturing
Prerequisite	Turning operation Avoidance dialog box® **Approach** group or **Departure** group
Location in dialog box	Turning operation dialog box® **Path Settings** group® **Non Cutting Moves** ® **Non Cutting Moves** dialog box® Approach page® **Approach Path** group or Departure page® **Departure Path** group

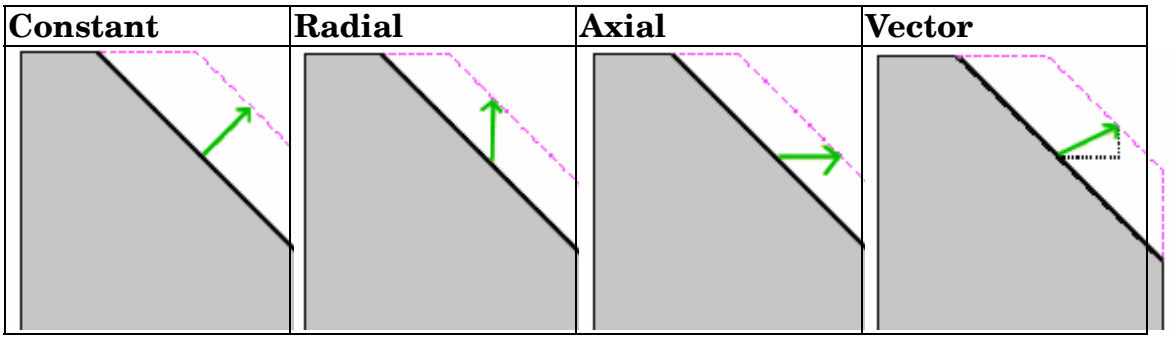
Custom boundary offsets

What is it?

In addition to the **Constant** custom boundary offsets, you can now apply translated offsets in an axial/radial direction or by a freely defined angle with:

- **Axial and Radial**

- **Angle and Distance**



Why should I use it?

Use this enhancement to define stocks and offsets which should not be applied perpendicularly to the boundary member.

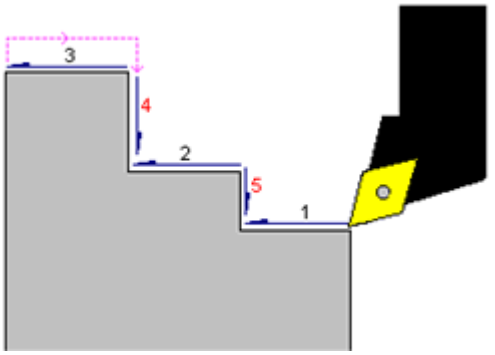
Where do I find it?

Application	Manufacturing
Prerequisite	Turning operation
Location in dialog box	Turning geometry dialog box® **Specify Part Boundaries**® **Part Boundaries** dialog box® **Edit**® **Edit Member** dialog box Main page® Select **Apply Member Stock**® **Stock Mode** Turning operation dialog box® **Custom Part Boundary Data** ® **Part Boundary** dialog box® **Edit**® **Edit Member** dialog box Main page® Select **Apply Member Stock**® **Stock Mode**

Finish operation cut order

What is it?

You can now control the cut order of multiple diameters and faces in a finish operation with the **Strategy** options **Diameters First, Then Faces** and **Faces First, Then Diameters**.



Diameters First, Then Faces

Why should I use it?

Use these options to avoid unnecessary transition moves between diameter and face cuts.

Where do I find it?

Application	Manufacturing
Prerequisite	Turning finishing operation
Location in dialog box	Finish operation dialog box® **Cut Strategy** group

IPW

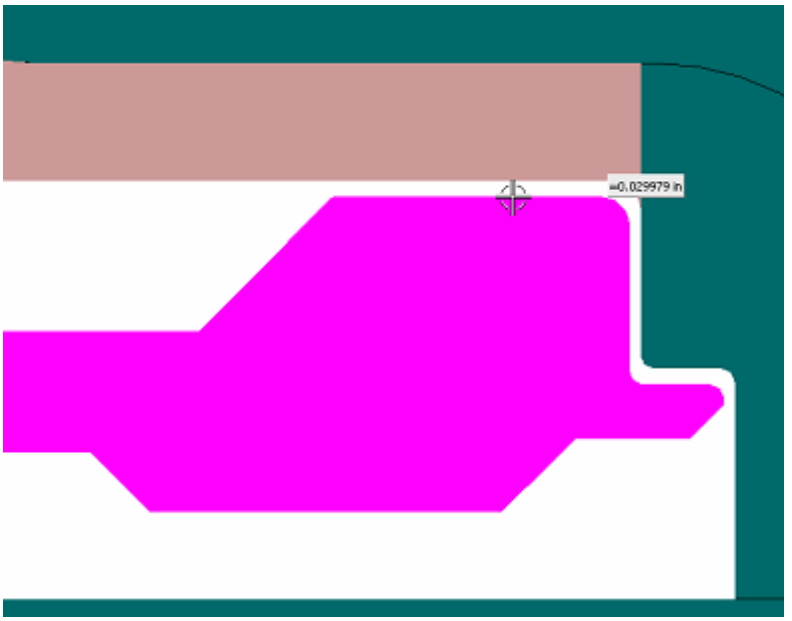
What is it?

The Turning In Process Workpiece (IPW) display now updates dynamically, to show the current condition of the part stock, when you select an operation in the **Operation Navigator**. You can select the 2D section curves of the IPW display to:

- Measure the distance from the IPW to the part geometry.
- Locate trim planes relative to the IPW.
- Locate positions relative to the IPW. For example, **Avoidance** geometry or **TeachMode** motions.

Note

Trim planes and other positions selected relative to the IPW are not associative to the IPW because the IPW display depends on the operation's tool path and is temporary.



Why should I use it?

With these enhancements you can:

- Determine the amount of stock remaining.
- Cut to the IPW edge instead of the part edge.
- Create an approach path that avoids the incoming IPW.
- Quickly determine the status of material removed, material left, or potential gouges by clicking on each operation.

Where do I find it?

Application	Manufacturing
Prerequisite	To display the turning In Process Workpiece (IPW), you must have a turning operation with an IPW generated and the **Display Turning IPW of selected objects** option turned on in **Customer Defaults** or **Manufacturing Preferences**.

See also:

- Operation Navigator customer defaults

Local returns and cycle interrupts

What is it?

Enhancements include:

- There are newly supported User Defined Events (UDE). You can combine the newly supported UDE with the traditional special events for local return.
- You can specify any of the supported UDE for cycle interrupts without generating a local return move.
- The sequence, content and NC output for legacy operations with local return events are preserved.

Why should I use it?

Use these enhancements for better machine control after a local return or cycle interrupt, or to output MCEs inside a cycle without moving the tool to a Local Return position.

Where do I find it?

Application	Manufacturing
Prerequisite	Turning operation
Location in dialog box	Turning operation dialog box® **Path Settings** group® **Non Cutting Moves** ® **Non Cutting Moves** dialog box® Local Return page

Feed rates

What is it?

You can now:

- Specify a different feed rate control for smaller cut segments of the part shape.
- Specify deceleration and acceleration feed rates for cutting into or away from a concave corner.

Where do I find it?

Application	Manufacturing

Location in dialog box	Turning operation dialog box® **Path settings** group® **Feeds and Speeds** ® **Feeds and Speeds** dialog box® **Short Segments** group or **Accelerate/Decelerate Limit** group Turning method dialog box® **Feeds** ® **Feeds** dialog box® **Short Segments** group or **Accelerate/Decelerate Limit** group

Probing operation

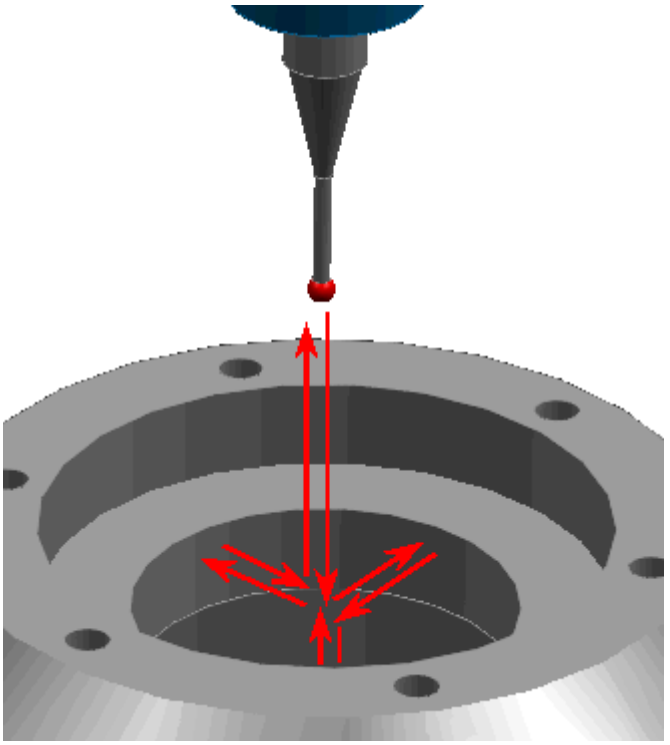
What is it?

The new **Probing** operation includes motions for positioning a single sensor tip. It supports the following Renishaw probing cycles:

- Calibrate probe length
- Calibrate stylus offsets and ball radius
- Calibrate on sphere
- Probe point
- Probe surface point
- Probe boss or bore

The Teamcenter and ASCII tool libraries support probe tools. The probe tools:

- Are solid models.
- Can be assemblies.
- Can include multiple tracking points for multiple sensor tips.



Where do I find it?

Application	Manufacturing
Toolbar	**Insert**® **Create Operation**
Menu	**Insert**® **Operation**
Prerequisite	**Probing** requires a solid probing tool and a separate license. **Type** group® **Probing**
Location in dialog box	**Operation Subtype** group® **Probing**

Feature based machining

Rule–based operations for features

What is it?

Rule–based operations for features is a proven technology seamlessly integrated into NX 6 that helps you automatically create operations such as milling, drilling and tapping. Rule–based operations:

- Let you select features such as holes, slots, and pockets from any source, including features that are User Defined, identified, recognized, or tagged.

- Apply best practice machining rules on the features while taking into account any defined PMI. A content kit with best practice machining rules is supplied with NX.

This important new approach for Feature Based Machining includes innovations in the following areas:

- Feature recognition
- Feature mapping
- Machining Knowledge Editor
- Standard machining content supplied with NX

Why should I use it?

Use rules–based operations to:

- Standardize on best practice machining knowledge. The software finds the best solution for your machining task within your company's environment.
- Save time with an automatic process.

Where do I find it?

Application	Manufacturing
Prerequisite	You must select **machining_knowledge** or a customized template from the Type list in the **Create Geometry** dialog box.
Menu	**Insert**® **Geometry** **Create Geometry**® **Geometry Subtype**
Location in dialog box	group® **FEATURE_PROCESS**
Shortcut menu	**Machining Feature Navigator** ® right–click in the background® **Identify CAD/UD Features** or **Recognize Features** **Machining Feature Navigator** ® select one or more features and right-click® **Create Geometry**

Feature recognition

What is it?

Enhancements to machining feature recognition include journaling and the recognition of:

- Diametrical and radial tolerances from PMI data.

- Surface finish from PMI data.
- Screw thread properties.
- Thread tolerances from PMI data
- 2½ D prismatic parameterized features such as slots and pockets.
- Complex stepped holes with up to four different diameters.
- Feature colors.

Why should I use it?

Use feature recognition to automatically create operations that use PMI data.

Where do I find it?

Application	Manufacturing
Shortcut menu	**Machining Feature Navigator**® right–click in the background® **Recognize Features**

Feature mapping

What is it?

Fully configurable functionality helps you map one machining feature to another. You can:

- Map the standard recognized feature types into your company's custom **User Defined Feature** types (UDF).
- Map your specific UDF feature types into the standard recognized feature types.

Why should I use it?

The mapping mechanism can serve to merge your company's existing FBM technologies with the new rule-based approach. You might map from one feature to another if:

- You have automatic machining process selection configured for your UDF and want to machine a part that was designed not using your company's standards.
- You would like to apply the standard automatic machining processes supplied with NX to your UDF.

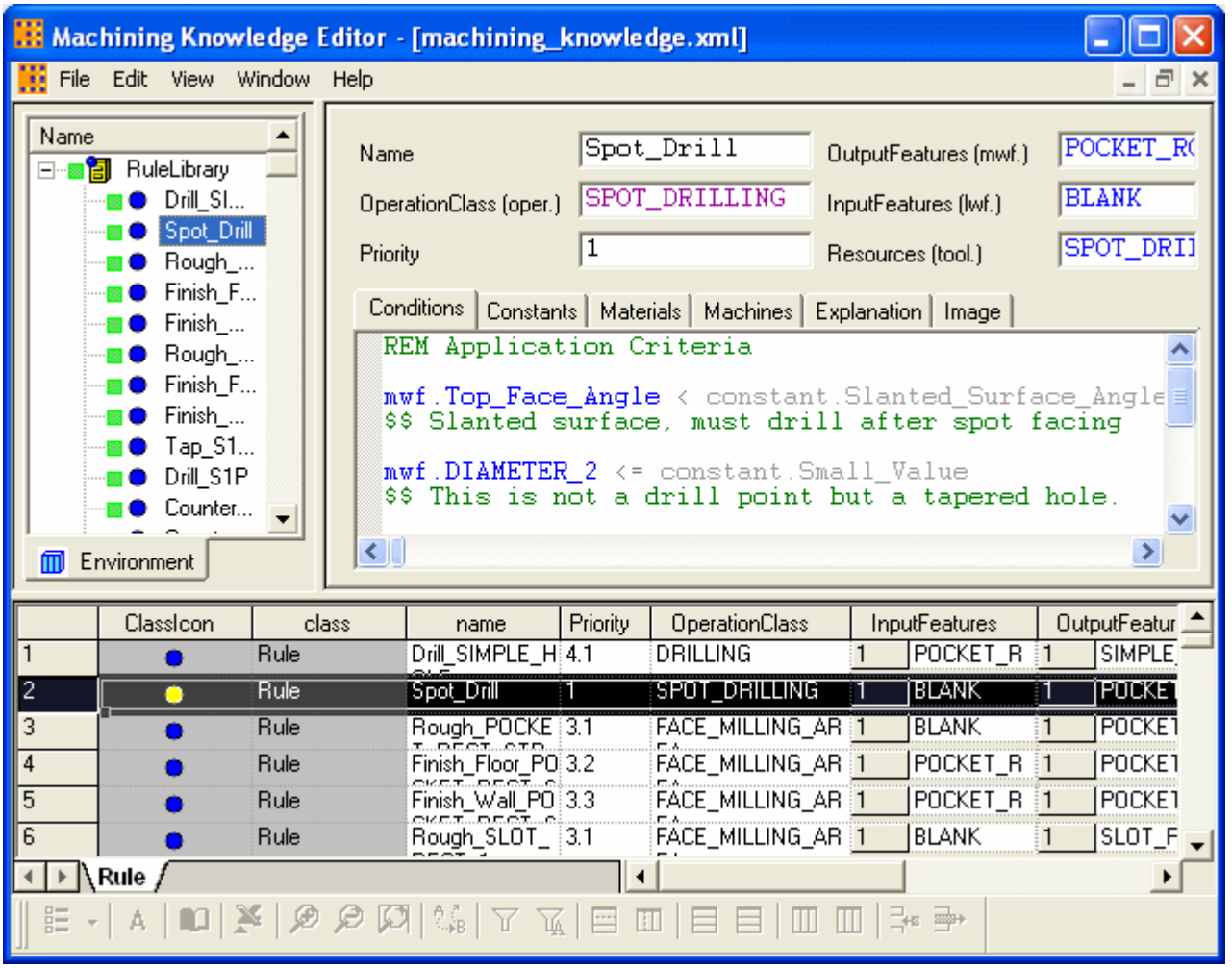
Where do I find it?

Application	Manufacturing
Shortcut menu	**Machining Feature Navigator**® right–click in the background® **Recognize Features**® **Feature Settings**® **Map Features**
File location	*\mach\resource\feature\feature_mapping_knowledge.xml*

Machining Knowledge Editor

What is it?

The **Machining Knowledge Editor** helps you create and modify the rule libraries which define the operations types and tools required to machine features.



Machining Knowledge Editor

Why should I use it?

Use the **Machining Knowledge Editor** to modify the standard machining knowledge supplied with NX or to define your company's best practices.

Where do I find it?

Application	Machining Knowledge Editor
Windows	**Start® All Programs® UGS NX 6.0® Manufacturing Tools® Machining Knowledge Editor**

Standard machining knowledge content supplied with NX

What is it?

The standard machining content supplied with NX includes best practice machining rules that:

- Apply to all recognized feature types.
- Use the tool types from the NX library.
- Are configurable with optional settings and threshold values.

Examples of feature mapping rules are also included in a separate file.

See also:

- Machining Knowledge Editor

Why should I use it?

Use the machining knowledge supplied with NX when you want to standardize the feature based machining process and when your company has not yet defined its own best practices.

Where do I find it?

Application	Machining Knowledge Editor
File location	*\mach\resource\machining_knowledge\machining_knowledge.xml*

Machining Feature Navigator filters

What is it?

Enhancements to **Machining Feature Navigator** filters let you:

- Filter on common attributes across multiple feature types.
- Edit existing filters.

- Use the **MCS** option to only list features that are parallel to the current tool axis.

Filters are now saved with the file to reduce space in the system registry and to make it easier to share filters with other users.

Where do I find it?

Application	Manufacturing
Shortcut menu	**Machining Feature Navigator**® right-click in the background® **Create/Edit Filters** or **MCS Scope**.

Collision Check and Incomplete status

What is it?

Enhancements to FBM include:

- Addition of **Collision Check** with a **Safe Distance** value.
- Display of an incomplete status by the **Operation Navigator** if the operation fails to generate the tool path for one or more features.

Why should I use it?

Use these enhancements to:

- Avoid collision with a part, such as a casting, that is not represented by the solid model.
- See which features were not generated so that you can apply different processes to them.

Where do I find it?

Application	Manufacturing
Shortcut menu	**Operation Navigator**® **Path** column

Wire EDM

What is it?

Wire EDM custom boundary data enhancements enable you to:

- Assign tab points anywhere on the boundary. Each tab is shown in the **Tab Points** list.
- Assign in-path events anywhere on the boundary, and assign any number of events at each location. Each event is shown in the **Events** list.

Events and tab points are displayed with a label in the graphics window. You can edit any tab point, event set, or location by selecting the list item.

Where do I find it?

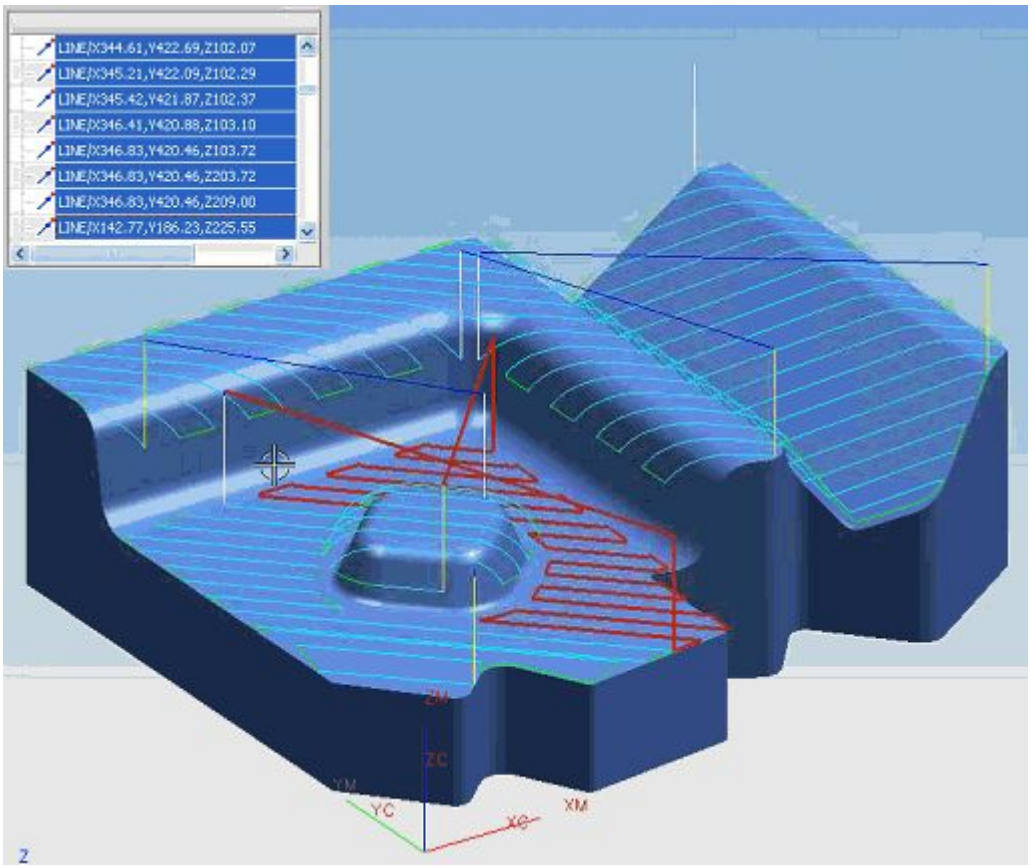
Application	Manufacturing
Prerequisite	Wire EDM operation
Location in dialog box	**Edit Geometry®** **Custom Boundary Data** **Custom Boundary Data** dialog box:**Tab Points** group and **Events** group

Tool Path Editor

What is it?

Enhancements include:

- The **Tool Path Editor** dialog box is completely redesigned, and is similar to the **Generic Motion** operation dialog box.
- The event list includes a **Feed Rate** column.
- You can select a range of motions from the graphics region or from the motion list.
- You can edit a selected event different ways: double-click it, right-click and choose **Edit**, or click .
- You can gouge check motions that have been edited.
- You can gouge check the entire path, a range of motions, or multiple motions that are not connected.
- You can reverse the entire path or a range of motions.
- When you select **Move**, there are additional options in the **Move** dialog box:
  - **Move Start Point** automatically moves the entire motion. You do not need to modify the start and end points individually.
  - **Transform Adjacent Circles** automatically modifies tangent circular motions to match your change. You do not need to manually reconnect these.



Why should I use it?

These enhancements make it easier to edit tool paths and give you more flexibility. For example, you can modify a range that spans between start and end events, or that contains a single region from a multiple depth operation.

Where do I find it?

Application	Manufacturing
Shortcut menu	**Operation Navigator**® right-click® **Tool Path**® **Edit**
Location in dialog box	**Tool Path Editor** dialog box® **Edit Actions** group

ISV

General

What is it?

Enhancements that apply to both the MTD and CSE controller models include:

- You can reload with linked posts. Postprocessing and simulation use kinematics information from the machine model to process the NC data. (Pivot point, gage point, rotary limits.)

- In many cases it is not necessary to add part geometry to the kinematic model. If there is no geometry in the kinematic model when ISV is opened, it looks in the operations and adds the appropriate geometry automatically.

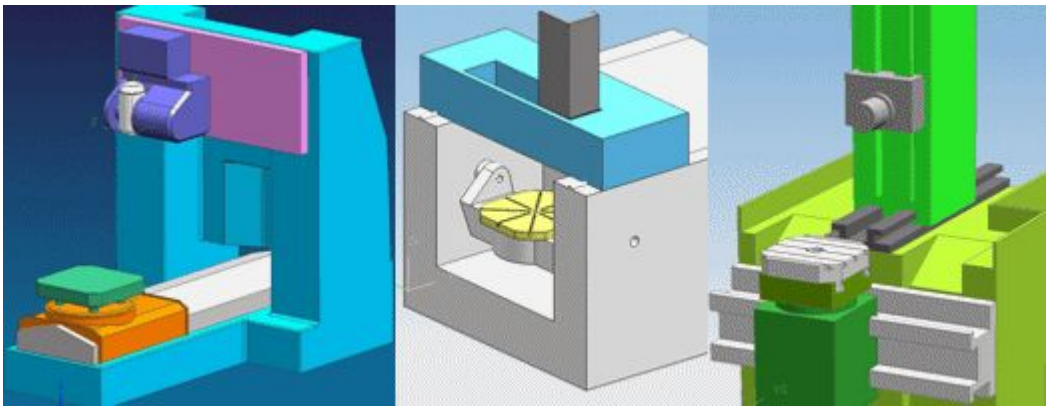
CSE

What is it?

CSE enhancements include:

- Support for a tilted MCS.
  - Siemens ROT, TRANS, FRAME
  - Fanuc G68, G68.1
  - Heidenhein Cycle19, G7
- Support for contact contour output.

  Contact contour uses the defined cutter for simulation. **CUTCOMP** has not changed.
- Support for the post to output tool tip coordinates with respect to the MCS.
  - Fanuc G43.1, G43.4 and G43.5
  - Siemens using TRAORI
- You can simulate one channel of a multi-channel machine.
- You do not have to step into subprograms. There is a new option to step over them.
- Improved structure of the mach kit sample machines and parts.
- A new set of sample machines with postprocessors and CSE simulation drivers that support Siemens, Fanuc, and Heidenhein controllers. All sample machines support the existing MCS. G54 code is not needed.



Collision checking

What is it?

Enhancements include:

- The IPW is used for collision checking.
- Hidden geometry is temporarily shown if it participates in a collision or limit violation.
- Collision checking is available against the tool holder.
- Collision checking is available between tools on different turrets.

Teamcenter Integration

Teamcenter Navigator with Manufacturing

What is it?

Teamcenter Navigator provides:

- Access to detailed information stored with operations in the Teamcenter database without leaving the NX application.
- An easy search capability.
- More load options available when you open a file.
- Advanced views that can reduce the level of complexity shown for the database.

Why should I use it?

Use **Teamcenter Navigator** to review NC documents and re-use CAM setups without leaving the NX application.

Where do I find it?

Resource bar	**Teamcenter Navigator**

Post Builder

UDE Editor

What is it?

The UDE Editor in Post Builder makes additional functions available in Post Builder to edit and create machine control events and user-defined cycles. The UDE Editor:

- Lets you interactively create, modify, and preview the definitions and event handlers for user-defined events and user-defined cycles.
- Creates a *CDL* file with the correct syntax for the definitions.
- Adds the event handlers to the *.tcl* file.

Why should I use it?

The **UDE Editor** makes it easier to create *CDL* files with the correct syntax, and to create event handlers without Tcl coding.

Where do I find it?

Application	Post Builder
Windows	**Start**® **All Programs**® **UGS NX 6.0**® **Manufacturing Tools**® **Post Builder**
Menu	**Options**® **Enable UDE Editor**® **Yes** (once for each post)
Location in dialog box	**Program & Tool Path** tab® **Program** tab **Machine Control** and **Canned Cycles** folders

Virtual N/C Controller (VNC)

What is it?

The following enhancements are made to the Virtual N/C Controller:

- The new Virtual N/C Controller page in Post Builder contains all the options required to interactively create machine-tool simulation drivers.

- There are many more graphical user-interface elements to help you create machine-tool simulation drivers.
- A VNC enabled post can also be prepared for standalone NC code simulation.

Why should I use it?

These enhancements make it easier to create *VNC* files with the correct Tcl syntax.

Where do I find it?

Application	Post Builder
Windows	**Start® All Programs® UGS NX 6.0® Manufacturing Tools® Post Builder**
Menu	**Program & Tool Path® Program**
Location in dialog box	**Virtual N/C Controller** tab

Localization

What is it?

Post Builder supports the following languages for the Windows platform:

- English
- French
- German
- Italian
- Japanese
- Korean
- Russian
- Simplified Chinese
- Traditional Chinese
- Spanish

You can switch to any of the installed languages before creating or opening a post. The language choice is remembered for the next session.

You can also install your own language file. For details, see the release notes.

Where do I find it?

Application	Post Builder
Menu	**Options® Language**

Other enhancements

What is it?

Other enhancements to Post Builder include the following:

- Formatted M and G code tables display the specified address leaders to better represent the NC instructions that are output.
- The **XZC-Mill** post is created with the **MOM_rotate** handler for the **Rotate** UDE.
- There are many new custom commands. See the release notes for details.
- The Post Files Preview page has been moved under the **Output Settings** group.

Windows only enhancements include the following:

- Mouse wheel support is available.
- You can link an external text editor (with the environment variable `EDITOR`) to the Custom Command page. (Post Builder 5.0.1)
- The Custom Command and File Preview pages have color coded text.

Where do I find it?

Application	Post Builder

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?

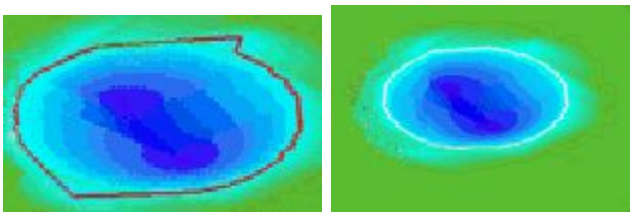
To test preview projects, contact the GTAC CAM support group with your request, and tell us why you are interested in the project. We will send you instructions for activating the project, logging incidents, and receiving support.

IPW thickness containment curves

What is it?

The following **Show Thickness by Color** options are only available for preview:

- Create containment curves that surround specified points to identify areas with the same amount of material remaining.
- Apply smoothing to the containment curves.



Why should I use it?

Use these **Show Thickness by Color** options to:

- Limit subsequent machining to areas with similar machining requirements.
- Improve containment curves with jagged edges to improve machining efficiency.

Where do I find it?

Application	Manufacturing
Prerequisite	A milling operation with tool path generated and an available 3D In Process Workpiece.
Resource bar	**Operation Navigator**® right-click the operation® **Tool Path**® **Verify** **Operation Navigator**® right-click the operation® **Workpiece**® **Show Thickness by Color** **Operation Navigator**® right-click the operation® **Tool Path**® **Simulate**

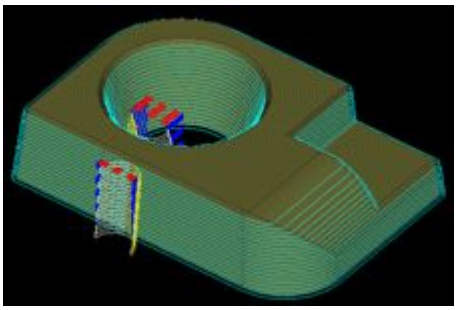
	Tool Path Visualization dialog box® **3D Dynamic** and **2D Dynamic** tabs
Location in dialog box	**Simulation Control Panel** dialog box® **Animation Settings** group

Divide by Holder

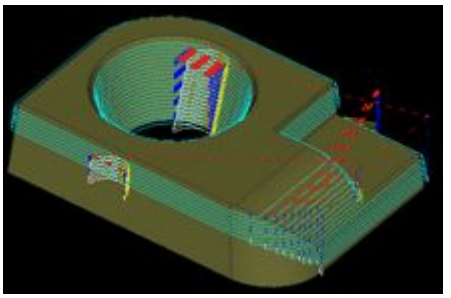
What is it?

When you use the holder to collision check an existing tool path, you can:

- Preserve the safe portions of the tool path.
- Save the gouging/colliding portions to a separate new operation and use a longer tool or a different tool axis.



Tool path with holder interference



Tool path after **Divide by Holder**

Why should I use it?

Use **Divide by Holder** to correct a tool path that was generated without considering the tool holder and length.

Where do I find it?

Application	Manufacturing
Prerequisite	A milling operation with tool path generated.
Resource bar	**Operation Navigator** ® right–click the operation and select **Tool Path**® **Divide by Holder**

Chapter

15 Automation

NX Open

Block Styler

What is it?

A new Block Styler application in NX provides block-based dialog box construction. NX Open developers can create NX dialog boxes that have the same user interface design as existing NX dialog boxes.

You can select from a catalog of blocks that support all aspects of NX interaction including data input, selection, expressions, and limits. Each of these blocks supports a list of properties that you can modify.

Where do I find it?

Application	Block Styler
Menu	**Start→All Applications→Block Styler**

Knowledge Fusion

KF TCE integration

What is it?

You can import and export DFAs and parts with DFAs into Teamcenter Integration. You also have load options for DFA files and KF applications. You can load the saved parts (having KF instances) in **As Saved** or **Use DFA Revision Rules** mode. You can also store the KF application related DFA files in TCE. Corresponding DLG files will still be stored in the application directory of KF application on the native file system.

Why should I use it?

You can manage DFA files in Teamcenter Enterprise.

Where do I find it?

Application	Teamcenter Enterprise **Tools® Knowledge Fusion® DFA Manager**
Menu	**File® Utilities® Customer Defaults® Knowledge Fusion**
Resource bar	**Knowledge Fusion Navigator®** click the root node and select **Add Child Rule**.
Command Line Utility	ug_clone

KF versioning and deprecation

What is it?

KF now provides application packaging versioning and deprecation control for classes, methods, and functions in the native NX environment.

Why should I use it?

Use the functionality to have better control over KF data and to better manage multiple revisions of KF data.

Where do I find it?

Application	Customer Defaults
Menu	**File→Utilities→Customer Defaults** **Knowledge Fusion→General→All→Load Application**
Location in dialog box	**Package / Native DFA Versions with Part® Use Latest Package or Version**

KF ICE best coding practices

What is it?

KF has implemented a smart code checker in ICE to verify the code against a list of best coding practices. Additionally, ICE provides automatic generation of documentation for KF classes, methods, and functions.

Where do I find it?

Application	**Class Editor**
Toolbar	**Knowledge Fusion® Class Editor**
Menu	**Tools® Knowledge Fusion® Class Editor**
Location in dialog box	In **ICE**, **File® Properties** and then on the **Syntax Check** tab on the **ICE Properties** dialog box

Knowledge Fusion optimization enhancement

What is it?

Optimization enhancements to KF classes include:

- The implementation of advanced optimization algorithms.
- The ability to allow optimization to be carried out in local as well as global domains for a given design.
- Multiple optimization problem solvers that cover a large range of optimization problems.
- The implementation of various solvers that provide global and local maximums and minimums, goal seeking activities, and multiple constraint handling.

Where do I find it?

Resource bar	**Knowledge Fusion Navigator**® right-click in the background® **Add Child Rule**
Location in dialog box	**Add Child Rule**® **nx_optimize**

KF ICE Application packaging

What is it?

A new **Application Explorer** tab in ICE shows KF application structures in hierarchical tree format. Additional functionality allows you to generate deployment-ready KF application packages in .zip or .tar file format.

Why should I use it?

Use the new functionality to streamline the KF application development and deployment tasks.

Where do I find it?

Application	**Class Editor**
Toolbar	**Knowledge Fusion**® **Class Editor**
Menu	**Tools**® **Knowledge Fusion**® **Class Editor**
Location in dialog box	**Application Explorer** page

Chapter

16 NX to JT

New configuration option

What is it?

A new **JtFileFormat** configuration option is added to the default JT configuration file *tessUG.config*. This option determines the JT file version.

If you want to write NX entities to version 9 JT files, you must set the value to 9. For example, JtFileFormat = “9”.

Version 9 JT files contain additional data recommended for viewing, for multi-CAD workflows and supplier exchange. They also enable more concise storage of tessellation.

> **Note**
>
> To be able to view version 9 JT files, you must have an appropriate viewer like Teamcenter 2007 or later. If you don’t have an appropriate viewer, use an earlier JT version, for example, “8”.

This option replaces the **mutiCADJT**, **compression**, **advCompression** and **advCompressionLevel** options.

Previously , you could write JT files to the required version using a combination of **compression** and **advCompression** options. Since you can now do so with just the **JtFileFormat** option, the compression-related options are not needed and are removed.

You can now apply Advanced Compression directly to individual tessellation LODs.

Discontinued configuration options

What is it?

The following configuration options are no longer available in the *tessUG.config* file:

- **multiCADJT**: This is replaced by the new **JtFileFormat** option.

- **UGtess**: Prior to the introduction of **XTbrep**, the **UGtess** command line option and the **UGtessallator** configuration option provided an alternative Parasolid tessellation for JT B-rep. Since this tessellation is achieved using **XTbrep**, the **UGtess** and **UGtessallator** options are no longer valid and are removed.

- **compression**: This option is removed since V8 JT onwards, all files are compressed by default.

- **advCompression**: Advanced Compression is applied individually to each tessellation LODs. For clarity, this option is removed from the EAI config section.

- **advCompressionLevel**: This option is moved to apply to the tessellation LOD section of the configuration file. This option is set for each individual LOD.

- **brepPrescision**: This option is always set to “DOUBLE” by default and has no configuration control.

- **checkFaceColors**: This option is removed as face colors are now processed by default.

Chapter

17 Translators

3D PMI export to 2D Exchange

What is it?

The 2D Exchange translator can now export 3D PMI from Modeling and the 3D PMI inherited on drawing sheets.

Why should I use it?

This functionality is useful if your NX part file contains 3D PMI information.

Where do I find it?

Application	Modeling / Drafting
Menu	**File→Export→2D Exchange**

3D PMI export to DXF and DWG files

What is it?

The DXF/DWG translator can now export 3D PMI inherited on drawing sheets to DXF and DWG files.

Why should I use it?

This functionality is useful if your NX drawing contains 3D PMI information.

Where do I find it?

Application	Drafting
Menu	**File→Export→DXF/DWG**

Chapter

18 Enhancements in NX 5.0.x Maintenance Releases

Teamcenter Integration

Authorized data access

What is it?

You can provide security for your CAD data with authorized data access. This enables access to sensitive or classified company data only by those users who have the appropriate privileges. Authorized data access applied to NX items in Teamcenter is also applied when those items are opened in NX.

Note

This functionality is available only with Teamcenter 2007.1.

Through the use of control mechanisms in Teamcenter, such as Access Manager, access to restricted and secure data can be defined and setup. In NX, the security designation is tied to the individual part and is passed to NX when the part is retrieved from Teamcenter.

Note

For information on setting up and configuring authorized data access in Teamcenter, including preferences and propagation of data access rules, see the *Teamcenter 2007 Security Administration Guide*.

The security of data in NX is controlled by:

- Logging and blocking
- Security access levels
- NX classification attributes

Logging and blocking

In NX, certain menu selections and actions are blocked for secure items and certain actions that could create a new copy of the item are logged to the Windows event log. NX does not propagate the secure designation to new files, therefore, the logging of actions and events provides an audit trail of what was done.

Note

Logging and blocking is optional.

To enable the blocking and logging functionality in NX, set the following preference in Teamcenter:

TC_session_clearance

Valid values: log, block, none

Log: Specific actions are logged to the Windows event log.

Block: Specific menu selections are blocked from being used in NX. This option also logs NX actions. NX Open programs and journaling that use the blocked functionality returns an error message.

None: (Default value) Logging and blocking are not applied.

For a listing of specific functionality that is impacted by logging and blocking, see the section NX functionality logged/blocked.

Security access levels

To define the list of clearance levels assigned to users and classification of data items, set the following preference in Teamcenter:

IP_level_list_ordering

Valid values: Strings on separate lines. The first line is the lowest classification level and the last line is the highest classification level. Multiple classifications can be on one line and are treated as equal.

User's access is determined by the classification of the part (ip_classification) and your clearance (ip_clearance). IP indicates intellectual property.

For example, values are:

```
confidential

secret

super-secret, top-secret
```

The following is applicable if the part (ip_classification) is secret:

- If your clearance (ip_clearance) is super-secret or top-secret, you can access the part and menus are not blocked.
- If your clearance is secret, you can access the part, but menus are blocked.
- If your clearance is confidential or you have no clearance set, you do not have access to the part.

Default values:

```
secret

super-secret, top-secret
```

Note

U.S. government data is designated with gov_classification. This is similar to the ip_classification and uses the preference ITAR_level_list_ordering to define clearance levels. In some cases, both ITAR and IP considerations are used. In these cases, ip_logged is set if either ip_classification or gov_classification are non-null and user_can_unmange can be derived from AM rules (where both ITAR and IP levels must pass). For additional information, see the *Teamcenter 2007 Security Administration Guide*.

NX classification attributes

Information about the classification status of a part and the user's privileges are mapped to attribute values in Teamcenter through attribute mapping. These mappings must be merged in with the other mappings for UGMASTER, UGPART, and DirectModel datasets. The attributes are as follows:

- **DB_PART_CLASSIFIED**

  Corresponds to the Teamcenter runtime attribute that identifies whether a part is classified by any mechanism. A mechanism can be just a non-null ip_classification value. Value is **T** (true) or **F** (false).

- **DB_USER_IS_PRIVILEGED**

  Corresponds to the Teamcenter runtime attribute that identifies whether the current user has access to all menus for this part. If false for any part in the session, the appropriate menus are blocked. Value is **T** (true) or **F** (false).

- **DB_PART_CLASSIFICATION**

  This attribute is for information purposes only and can be used in an Assembly Navigator column. Value is the value of the ip_classification property for the dataset/owning item in Teamcenter.

An example of attribute mapping is as follows:

```
{ Dataset type="UGMASTER"
# (hard-wired) DB_PART_NAME: "Part Name"
# (hard-wired) DB_PART_DESC: "Part Description"
DB_USER_IS_PRIVILEGED: user_can_unmanage /master=iman /description="User privileg
DB_PART_IS_CLASSIFIED: ip_logged /master=iman /description="Part logging requirem
DB_PART_CLASSIFICATION: ip_classification /master=iman /description="IP classific
```

NX functionality logged/blocked

The following NX functions/operations are impacted when secure data is accessed. If your security clearance is equal to or less than the part classification, the function/operation is blocked. Those identified as logged are always logged.

> **Note**
>
> If your security clearance is greater than the part classification, you are not blocked and the function/operation is just logged.

The following NX functionality is logged:

- Part family update
- Substitute component
- Create new parent
- Paste a component
- KF Part family update
- Drag a sheet template onto a part

The following NX functionality is blocked:

- Save outside of Teamcenter
- Export Catia V5
- High quality image plot
- High quality image save
- View animation
- View navigate
- Export IGES
- Export Step203

- Export 2D exchange
- Part family create
- File->Interoperate
- File->Collaborate
- Print
- Export PDF
- Assembly Navigator export to browser
- Export user defined feature
- Plot
- Export Step214
- Export DXFDWG
- Export CGM
- Add existing component
- Create new component
- Save as
- Export polygon file
- Export V4 Catia
- Export parasolid
- KF Save part as
- KF Create new part
- Export drawings to Teamcenter
- Assemblies create clone
- Author HTML
- Send to package file
- Export STL

- Export VRML
- Export part
- Export HTML
- Spreadsheet save as

Why should I use it?

You can access secure CAD data in NX.

Where do I find it?

Secure items are automatically designated as such in NX once the functionality is setup in Teamcenter and NX..

Advanced Search

What is it?

Advanced search allows you to search Teamcenter from within NX using any pre-defined Teamcenter query. There are several queries provided by default, and you can configure any Teamcenter query so it is available.

Note

This functionality is available with Teamcenter 2005 SR1 MP3.

The **Advanced Search** dialog box contains the components for conducting an advanced search.

The functionality includes a **Look for** list from which you select the type of query to perform, such as searching for **Any Object**, or **Item**, or **Model**. By selecting the correct query type you can narrow your search to a manageable list of items. The provided query types are:

- Any Object (the default)
- Model
- Assembly
- Drawing
- CAM
- Simulation
- FEM

- Multi-CAD
- Alt Rep
- Item
- General
- Checked Out Data Set
- Item rev

Note

Some of these are Teamcenter provided queries.

The search criteria changes depending upon the type of query selected. For example, different search criteria are presented for **Item** than for **All Objects**. Options for specific criteria also change depending upon the criteria type selected. For example, **Date Criteria** of **Last Modified** displays the option **Date Type**.

The search is conducted in the Teamcenter database but only those results that can be handled by NX are listed, such as NX parts, assemblies, and so on. Folders are not returned by the search.

The last search performed is shown as a **Last Search** folder and contains the items found during the search. Click **Save As** to save the search results to a saved search folder. You can open a saved search folder or **Last Search** folder, redefine the search criteria, and rerun the search on those items in the folder.

Note

The ability to save the search results to a folder is available with Teamcenter 2007.

You can set the number of results to display in the **Advanced Search** dialog box by setting the customer default. Choose **File→Utilities→Customer Defaults**. From the **Teamcenter Integration for NX** list, select **General**, and on the **General** tab, set the following default:

Number of results to display

To enable Teamcenter provided queries for display in the **Advanced Search** dialog, edit the following Teamcenter preference:

NX_exposed_queries

Why should I use it?

Use **Advanced Search** to perform more precise searches.

Where do I find it?

Choose **Tools→Teamcenter Integration→Advanced Search**.

This functionality is also available in **Teamcenter Navigator** and in any file selection dialog box.

Cloning multi-CAD items

What is it?

Multi-CAD items can be cloned. You can make a complete copy of an assembly that contains multi-CAD items. They are handled the same as NX items in an assembly when you create a new cloned assembly or edit an existing cloned assembly.

Note

Multi-CAD items consist of non-NX items, such as JT and I-deas datasets.

Multi-CAD items are included in the same functionality as NX items during cloning. For example, when you click **Add Assembly** in the **Clone Assembly** dialog box, multi-CAD items are listed with the NX items.

Note

The log file generated during cloning is not attached to the cloned multi-CAD dataset as a named reference as it is with NX items.

The multi-CAD capability is also included in the following utility programs when you add the following options:

Utility	Options
ug_clone	–pim=yes –o=clone
ug_clone	–pim=yes –o=edit
ugmanager_clone	–o=clone
ugmanager_clone	–o=edit

Teamcenter Business Modeler Deep Copy Rules establish rules to copy forward primary datasets that are associated with a specific item revision type. The **Deep Copy Rules** must be set to enable copying of the datasets that are to be cloned.

The multi-CAD functionality is not available for the clone based import and export assembly commands (**Tools→Teamcenter Integration→Import Assembly, Export Assembly**).

Why should I use it?

The capability enables you to make a clone of an assembly that contains multi-CAD items.

Where do I find it?

Choose **Assemblies→Cloning→Create Clone Assembly**.

Choose **Assemblies→Cloning→Edit Existing Assembly**.

Replacement component in an assembly retains data

What is it?

When you replace a component in an assembly with the **Component As** command, the component retains the characteristics provided in Teamcenter. The occurrence notes and variant information provided in Teamcenter PSE for the replacement component are kept upon substitution.

You can also replace a component in an assembly regardless of where it is used in the assembly.

Note

This does not change the functionality of the **Assembly As** command.

Where do I find it?

In the **Assembly Navigator**, right-click **Open** and choose **Component As**.

Fully load data for current version of a part

What is it?

The customer default **Always fully open component from original version** loads data from the original version of the part. When there is a need to load more data for a part which is partially loaded in a session, the data is loaded from the same version of the part that is currently loaded, even if a newer version of the part is available.

If the same version as the currently loaded part is not available, such as due to version limits, the operation is cancelled and you have to close and reopen a newer version of the part.

If a later version of the part exists but the data is loaded from an earlier version, you get a warning message when you:

- Set the part as the work part or displayed part
- Modify the part

- Use the **Save** or **Save As** commands on the part

You cannot save changes to the part if a later version already exists in the database. You must perform a **Save As** to save the changes to a new Item (not a new Item Revision).

Why should I use it?

You get all the data from the currently loaded version of the part you are using.

Where do I find it?

This functionality occurs whenever additional data is necessary depending upon the operations being performed in NX.

Choose **File→Utilities→Customer Defaults**.

From the **Teamcenter Integration for NX** list, select **General**.

Click the **Assembly** tab.

Open Part File and Teamcenter Navigator columns are configurable

What is it?

The display of the columns in the **Open Part File** dialog box and **Teamcenter Navigator** are configurable. You can specify the columns that are displayed and change the order in which they appear.

In the **Open Part File** dialog box or **Teamcenter Navigator**, right-click on a column heading and select **Columns→Configure**. In the **Columns** dialog box, you can turn the display of a column heading on or off with the checkbox, or move the location of a column heading with the up and down arrows.

Note

The first column heading named **Object** cannot be turned off or moved.

Why should I use it?

You can customize the display and location of the **Open Part File** and **Teamcenter Navigator** column headings.

Where do I find it?

In the **Open Part File** dialog box or **Teamcenter Navigator**, right-click a column heading and choose **Columns→Configure**.

NX Essentials

Command Finder

What is it?

Command Finder is a new search tool which helps you find and activate a specific NX command that is associated with one or more given words or phrases. This includes commands that may not be active in the current application or task environment.

> **Note**
>
> Results are limited to commands available in the main menus or on toolbars. Commands contained only on background shortcut menus or in navigators are not included in the search.

From the list of commands you can:

- Display the command location, when it is available in the current environment.
- Launch the command, if it is available.
- Turn on a toggle command, when it is available in the current environment.
- Access the Help information for the command.

Options for **Command Finder** are located on the **Command Finder** tab in the **File→Utilities→Customer Defaults→Gateway→Extras** dialog box. These options allow you 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?

- On the **Standard** toolbar, click **Command Finder** .
- Choose **Help→Command Finder.**

Default names for new groups

What is it?

The **New Group** command assigns a default name to each new group you create. The name includes a number that increments with each new group. For example, **Group_1**, **Group_2**, **Group_3**, and so on.

Where do I find it?

Choose **Format® Group® New Group**.

Record and Play C# Journals

What is it?

In addition to previous support for VB.NET, C++ and Java, C# is now available for recording journals. Before you can record a journal using C#, you must set the appropriate journaling language preference.

You can also replay a C# journal. Previously only journals recorded in VB.NET were supported for replay. Now, when a journal with either a .vb or a .cs file extension is selected from the **Journal Manager**, the **Run** button becomes available. If either type of journal is loaded in the **Journal Editor**, the **Play** button becomes available.

Where do I find it?

To set the journaling language preference:

Choose **Preferences** → **User Interface**. In the **User Interface Preferences** dialog box, click the **Journal** tab, and then select **C#** under **Journal Language**.

To record a journal, do one of the following:

- Choose **Tools→Journal→Record** .
- On the **Journal** toolbar, click **Record** .

To play a journal, do one of the following:

- Choose **Tools** → **Journal** → **Play**.
- In the **Journal Manager** dialog box, click the **Run** button.
- In the **Journal Editor** dialog box, click **Play**.

Select Scope in Sketcher

What is it?

In the Sketcher, you can now change the **Select Scope** option while a command dialog box is open.

Once you set the selection scope for a particular command, NX remembers it throughout the current session.

Where do I find it?

In the Sketcher, open a command dialog box and on the Selection Bar, click **General Selection Filters** to see **Select Scope**.

Print dialog box

What is it?

The **Print** dialog box now provides the option to print any drawing sheet contained within the current part file, as well as options to control the thick, thin, and normal line widths used to print selected drawing sheets.

These options are also available in the **Customer Defaults** dialog box, along with options for output type and background setting.

The **Print** command can also now be recorded in a journal and replayed.

Note

The **Print** command is available for Windows platforms only.

Where do I find it?

- Choose **File→Print**.
- Choose **File→Utiltiies→Customer Defaults**. From the **Gateway** list, select **Extras**. Click the **Printing** tab.

Infer Edge Output

What is it?

Infer Edge Output simplifies preparation of plot-type output for **Plot** (**File® Plot**), **Print** (**File® Print**), **export CGM**, (**File® Export® CGM**), and similar commands.

When NX plots a modeling view, it uses the view in the graphics window instead of the actual **Static Wireframe** settings for the view. This affects plot-type output for modeling views only; output for views on a drawing sheet is not affected.

Why should I use it?

If you like to use the **Wireframe with Hidden Edges** or **Wireframe with Dim Edges** rendering styles, you can quickly determine the appearance of your plot output by looking at the view in the graphics window. This eliminates confusion, as you do not have to worry about how the **Static Wireframe** options are set.

Where do I find it?

1. Choose **Preferences® Visualization**.
2. Click the **Visual** tab
3. In the **Edge Display Settings** group, under **Session Settings**, click the **Infer Edge Output** check box.

Design

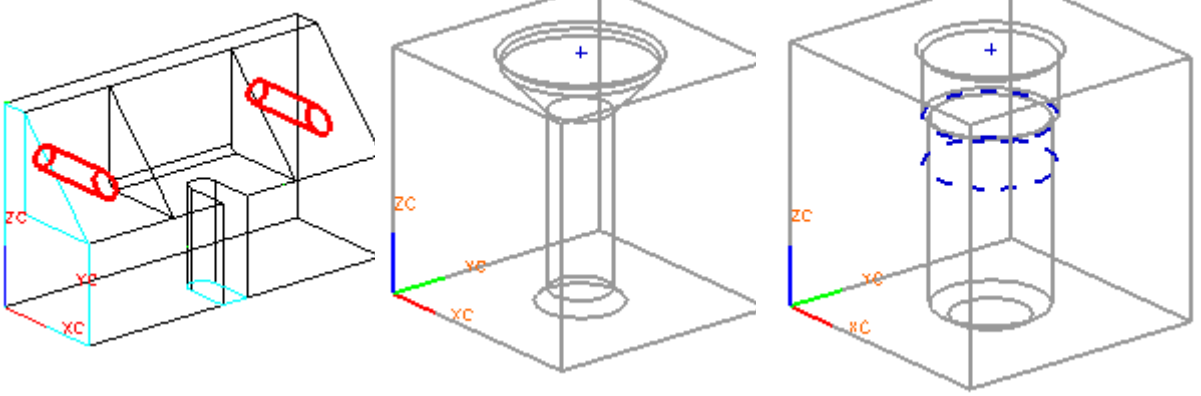
Modeling

Hole

What is it?

You can now use the **Hole** command to create counterbored or countersunk holes of the following types:

- General holes
- Screw Clearance holes
- Threaded holes



General Hole **Screw Clearance Hole** **Threaded Hole**

You can:

- Create holes on non-planar faces.
- Create multiple Hole features by specifying multiple placement points.
- Specify the position of the Hole feature using Sketcher. Snap Point, and Selection Intent options are also available to aid selection of existing points or feature points.
- Create Hole features using formatted data tables for the **Screw Clearance** and **Threaded Hole** types.
- Use standards like ANSI, ISO, DIN, JIS, GB, and so on.
- Use the **None** and **Subtract** Boolean commands on the tool bodies while creating a Hole feature.
- Optionally add start, end, or relief chamfers to the Hole feature.

Where do I find it?

In the Modeling application:

- Choose **Insert**® **Design Feature**® **Hole**.
- On the **Feature** toolbar, click **Hole** .

Hole Series

What is it?

Hole Series is a new Hole type in the **Hole** command.

You can use **Hole Series** to create a series of linked holes in assemblies that have:

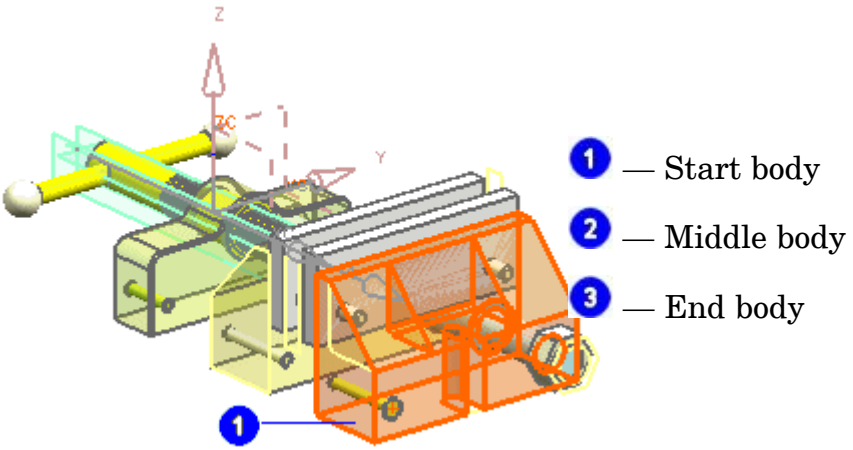
- Multiple bodies in the work part.
- Multiple components within a single body.
- Multiple components that contain multiple bodies.

Hole Series provides a set of related holes needed to mount a fastener across multiple components. The target components need not be children of the work part.

When you specify the origin and the direction of the hole, depending on the customer default settings, NX infers the start, middle and end bodies within the assembly or the work part that the hole form will intersect.

You can:

- Edit the inferred selection of the start, middle and end bodies.
- Match the middle and end holes to the start hole.
- Specify the form and dimensions of the start, middle and end holes independently.



Note

If you directly edit a hole that is a child of a Hole Series, the associative link of the hole series is broken and the hole is converted to a stand-alone Hole feature. The rest of the series is unaffected. However, you cannot recreate the link to the parent.

Why should I use it?

Use the **Hole Series** type of Hole feature to create multi-form, multi-target body, concentric holes with coordinated dimensions.

Where do I find it?

Application	Modeling
Toolbar	**Feature® Hole**
Menu	**Insert® Design Feature® Hole**
Location in dialog box	**Type** group® Type list® **Hole Series**

Customer defaults for **Hole Series**

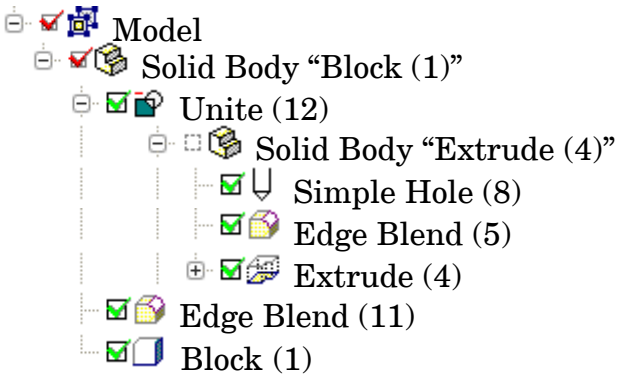
Menu	**File® Utilities® Customer Defaults**
Location in dialog box	**Modeling® Extras®** Hole Series page

Bodies and Booleans

What is it?

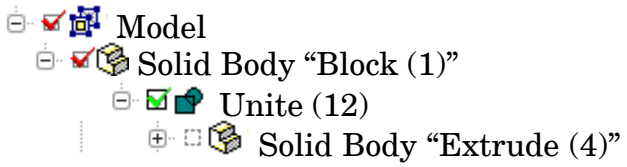
A **Bodies and Booleans** command is now available in the **Part Navigator** shortcut menu for solid bodies.

You can use this option to view the Booleans of a selected solid body. This option is available only when **Timestamp Order** is turned off.



Part Navigator — main panel with Timestamp Order turned off

If you right-click a body that has Booleans, and choose the **Bodies and Booleans** command, the **Part Navigator** shows the Boolean features of the selected body.



Part Navigator — Bodies and Booleans view

Why should I use it?

You can use this option to view the tool body and Boolean features of a selected body in the **Part Navigator**.

The view of the history tree is simpler, with more pronounced branches, which makes it easier to view the Booleans of the selected solid body.

Where do I find it?

1. On the Resource Bar, click the **Part Navigator** tab.

2. Right-click in the background of the **Part Navigator** and clear the **Timestamp Order** check box if it is selected.

3. In the main panel, right-click a solid body node and choose **Bodies and Booleans**.

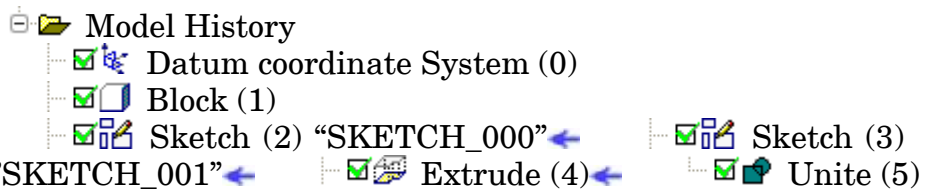
Select Tool Features

What is it?

You can now use the **Select Tool Features** command to select reference features (such as sketches, datums, and curve features) of the tool body of the selected Boolean.

A new customer default option **Also Select Reference Features** controls whether reference features are to be selected. The check box is selected by default. When you clear this check box, **Select Tool Features** selects only those features which are directly related to the tool body of the Boolean.

In the main panel of the **Part Navigator**, if you right-click **Unite (5)** and choose **Select Tool Features** from the shortcut menu, the tool body **Extrude (4)** and the reference features **Sketch (2) "SKETCH_000"**, **Sketch (3) "SKETCH_001"** are selected.



Part Navigator main panel view

Why should I use it?

This enhancement lets you select more features related to the references of the selected tool body. It also minimizes the task of resolving any external references when the selected features are copied and pasted.

Where do I find it?

To find the **Select Tool Features** command:

1. On the Resource Bar, click the **Part Navigator** tab.

2. In the main panel of the **Part Navigator**, right-click a Boolean node.

To find the **Also Select Reference Features** customer default option:

1. Choose **File® Utilities® Customer Defaults**.

2. Select **Modeling® Extras**.

Show and Hide

What is it?

Show and **Hide** commands have the following enhancements:

- **Show/Hide** and related commands are now available in a dedicated category in the shortcut menu for selected objects.
- These commands are now available on the **Part Navigator** shortcut menu even if the selected objects are a mix of objects that are shown or hidden.
- When you use the **Show** command on an object on an invisible layer, you can either have the object move to the work layer or make the invisible layer selectable. The default action depends on the customer default settings. Select either the **Move to Work Layer** or the **Change Layer to be Selectable** customer default option.
- **Hide Body** is now called **Hide**.
- **Show Parents** and **Hide Parents** commands are now available in all NX applications.

Where do I find it?

To see the **Show/Hide** or **Show Parents/Hide Parents** shortcut menu commands:

- Right-click an object node in the main panel of the **Part Navigator**.
- Right-click an object node in the graphics window.

To see the **Move to Work Layer** and **Change Layer to be Selectable** customer default options:

1. Choose **File**® **Utilities**® **Customer Defaults**.
2. Select **Gateway**® **Part Navigator**.
3. The options are available in the **Parents on Invisible Layer – Action on Show** section.

Direct Modeling

Direct Modeling commands are used to modify a model regardless of its origins, associativity, or feature history. The model could be imported, non-associative, with no features, or a native NX model complete with features.

Direct Modeling is primarily suited for use on models composed of analytic faces like plane, cylinder, cone, sphere, torus. This does not necessarily mean

simple parts, since models with many thousands of faces are composed of these face types.

The following Direct Modeling commands are now enhanced.

Direct Modeling command	What's new
Resize Face	New design.
Offset Region	Now supported by Selection Intent and dimension handles.
Replace Face	Now supported by Selection Intent.
Move Face	Move Region is replaced by Move Face.
Pattern Face	Now supported by Selection Intent and dimension handles.
Resize Blend	Reblend Face is replaced by Resize Blend.
Delete Face	New command.

Move Face

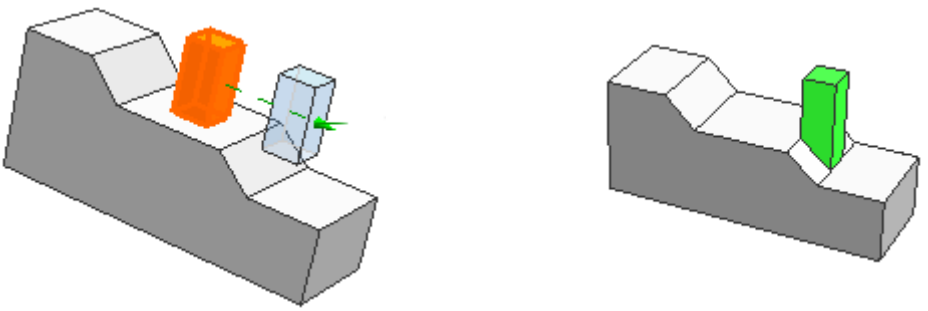
What is it?

Move Region is now called **Move Face**. The command has the following enhancements:

- You can now select faces to move using Selection Intent.
- You can set the distance or angle using dimension handles.

> **Note**
>
> The **Move Face** command that was previously available from the **Edit** menu (**Edit**® **Face**) is now removed.

In the following example, a face is moved using **Move Face**.



Why should I use it?

Use **Move Face** to move one or more faces on a body.

Where do I find it?

In the Modeling application:

- Choose **Insert**® **Direct Modeling**® **Move Face**.
- On the **Direct Modeling** toolbar, click **Move Face** .

Delete Face

What is it?

Delete Face is a new Direct Modeling command that enables you to delete faces from a model and heals the open area left in the model by the deleted face, by extending adjacent faces.

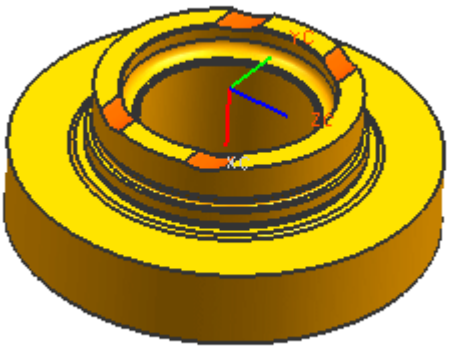
Why should I use it?

Delete Face is especially useful when modifying an imported model which has no feature history. A model created in NX has feature history, and you can use the **Part Navigator** to delete features and remove unwanted geometry from the model.

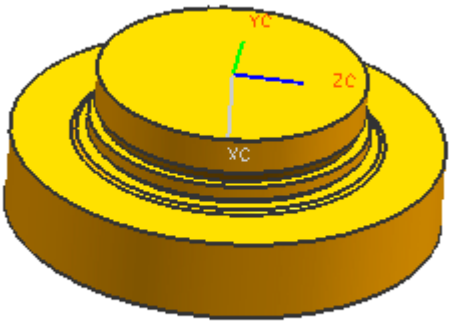
After you delete a face, the delete face feature appears in the history of the model. You can edit or delete this like any other feature.

The following is an example of how a model heals after using **Delete Face**.

Before using **Delete Face**



After using **Delete Face**



Where do I find it?

In the Modeling application:

- Choose **Insert**® **Direct Modeling**® **Delete Face**.

- On the **Direct Modeling** toolbar, click .

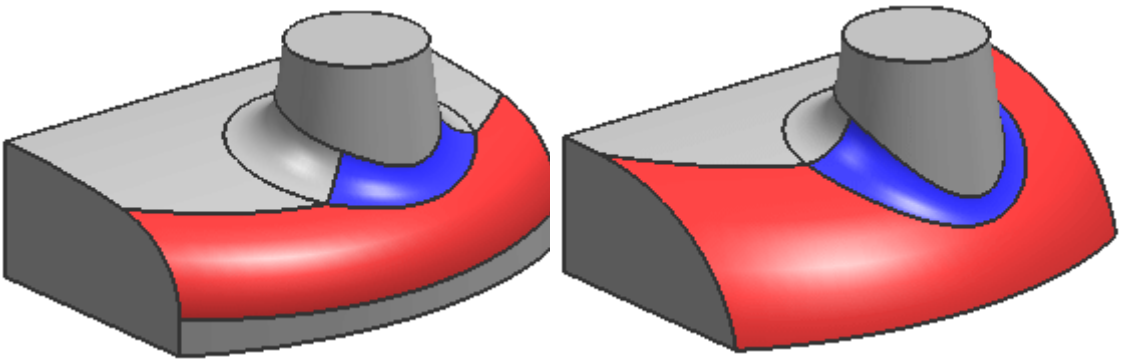
Resize Blend

What is it?

Reblend Face is now called **Resize Blend**. The command has the following enhancements:

- When you change the size of a blend, dependent blends are updated.
- You can select faces to resize using Selection Intent.

In the following example, the red face in the body on the left is resized using **Resize Blend**. The dependent blue face updates automatically.



Why should I use it?

Use **Resize Blend** to edit the radii of blend faces, regardless of their feature history.

Where do I find it?

In the Modeling application:

- Choose **Insert**® **Direct Modeling**® **Resize Blend**.
- On the **Direct Modeling** toolbar, click **Resize Blend** .

Assemblies

I-deas migration to assembly constraints

What is it?

You can now migrate I-deas assembly constraints directly to NX assembly constraints, instead of to mating conditions.

Why should I use it?

An assembly migrated from I-deas with NX assembly constraints is more similar to the original I-deas assembly. This is because I-deas assembly constraints are more similar to the NX assembly constraints than to NX mating conditions.

Assembly constraints also have more functionality than mating conditions. See the *Assemblies Help* for more information.

Where do I find it?

1. Choose **Preferences**® **Assemblies** to open the **Assembly Preferences** dialog box.
2. Set **Assembly Positioning**® **Interaction** to **Positioning Constraints**.
3. Migrate an I-deas assembly.

Move Component types

What is it?

You can move components with the following methods that are now available on the **Type** menu of the **Move Component** dialog box:

- **Between Two Points**
- **Along Vector**
- **Rotate about Axis**
- **Between Two Axes**
- **Reposition**
- **Rotate Using Points**

These new options correspond to methods that are currently available on the **Reposition Component** dialog box, which **Move Component** will eventually replace.

Where do I find it?

Choose **Assemblies**® **Components**® **Move Component**.

Mirror Assemblies Reposition

What is it?

You can use the new **Reposition** button on the Mirror Review page of the **Mirror Assemblies** wizard to change the mirror type from **Mirror Geometry** to **Reposition**. **Mirror Geometry** creates a new component part file, while **Reposition** does a two-dimensional repositioning of a new instance of a selected component.

After you click **Reposition**, you can cycle through the possible positioning solutions by clicking the **Cycle Mirror Solutions** button.

Why should I use it?

You can now easily change the mirror type to the **Reposition** type before you create the mirror assembly.

Where do I find it?

Choose **Assemblies**® **Components**® **Mirror Assembly**.

Linked sketches in the WAVE Geometry Linker

What is it?

You can link a sketch from one part of your assembly to the work part with the new **Sketch** option on the **Type** menu in the **WAVE Geometry Linker** dialog box.

Why should I use it?

Sketch lets you select an entire sketch for WAVE linking with a single click.

Where do I find it?

Choose **Insert**® **Associative Copy**® **WAVE Geometry Linker**.

Documentation

PMI

Import Model Views and PMI from JT files

What is it?

You can now import model views, view sets, and PMI symbols defined in the model views from .jt files.

Why should I use it?

This enhancement allows a .jt file opened in NX to more closely resemble the original file, as it:

- Increases the types of information contained in the .jt file.
- Provides better management of PMI in the .jt file.

Where do I find it?

1. Choose **File® Open**.
2. In the **Open Part File** dialog box, change **Files of type** to **JT Files (*.jt)**.
3. For the **File name**, select a .jt file that contains model views and PMI symbols.

Data Reuse

Teamcenter Classification objects in the Reuse Library

What is it?

Enhancements to the **Reuse Library** enable more thorough browsing of the Teamcenter Classification hierarchy tree, and provide direct access to classified objects.

As you browse the Teamcenter Classification hierarchy tree, you can drag any classification object that has a relevant NX part associated with it into the graphics window.

Use the **Tree Search** group to find a particular class node in the classification hierarchy. Use the class attribute search capability to find a particular class member within a class.

Why should I use it?

Accessing Teamcenter Classification from within NX streamlines the design process.

Where do I find it?

The **Reuse Library** tab is on the **Resource Bar**.

Machinery Library installation tool

What is it?

The **NX Machinery Library** includes an extensive set of industry standard parts. It is delivered as a series of ZIP files, each file representing a particular global standard i.e. one ZIP file for ANSI inch, another for DIN, and so on. The new installation tool guides you through the install process to a folder you specify in either the Native NXor Teamcenter Integration environment. On each page you can specify the installation language, installation environment and the specific libraries to install. You can also configure the industry standard parts to your specifications.

Why should I use it?

Because the installation tool highlights each step of the install process in its wizard-like interface, you are less likely to miss a critical step.

Where do I find it?

The **NX Machinery Library** and the installation tool are available to any customer with an NX Mach license under maintenance. Download them through the Full Product Download section of the UGS Customer FTP site.

Note

You are not required to download or use the **NX Machinery Library**.

NX Machinery Library

What is it?

The NX Machinery Library includes an extensive set of industry standard parts delivered as a series of ZIP files. Each ZIP file represents one global standard, for example, ANSI, DIN, and so on.

Why should I use it?

Newer versions of the NX Machinery Library contain new and revised part families. You can install the newer library over an old installation.

Where do I find it?

The NX Machinery Library and the installation tool are available to any customer with an NX Mach license under maintenance. Download them through the Full Product Download section of the UGS Customer FTP site.

Note

You are not required to download or use the NX Machinery Library.

System Design

Die Engineering

Addendum Section

What is it?

Addendum Section has the following improvements:

- You can now define one or more addendum sections and use them to create an addendum surface later.

- You can create sections after selecting points and edit parameters as soon as sections are created.

  Each section lies in a plane you define and is composed of eleven individual segments, some of which may collapse to zero. You can control the shape and orientation of each segment through the available parameters.

- You can select edges to define the span of the preview surface. Selected edges are connected and the first and last points of the connected string are used for the limit points.

- You can use the **Edit** option to manually alter the parameters to define the shape of the section. When you change a parameter, the software automatically adjusts other parameters of the curve to ensure that the section meets formability rules.

Why should I use it?

Use the **Addendum Section** command to create an addendum section which defines the shape of the addendum surface.

Where do I find it?

In the Modeling application:

- Choose **Tools® Vehicle Manufacturing Automation® Die Engineering® Addendum Section**.

- On the **Die Engineering** toolbar, click **Addendum Section** .

Draw Bead

What is it?

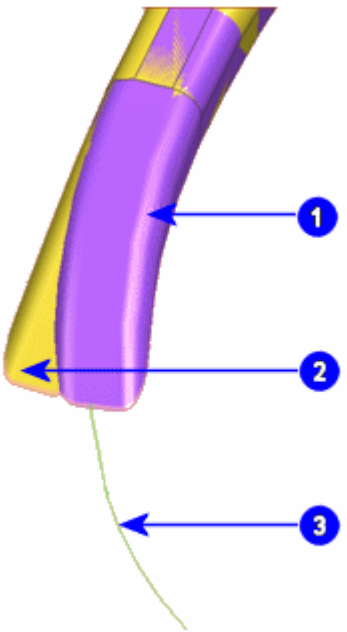
Use the **Draw Bead** command to apply a bead shape to a sheet body. You can also create two sheets representing the face of the upper and lower die containing the male and female bead shapes.

The **Draw Bead** command has the following enhancements:

- You can create flow beads of depth less than the radius.

- You can specify the female bead width (W) as a function: W = MW+ 2(t+L), where MW is the male bead width, t is the material thickness and L the clearance values.

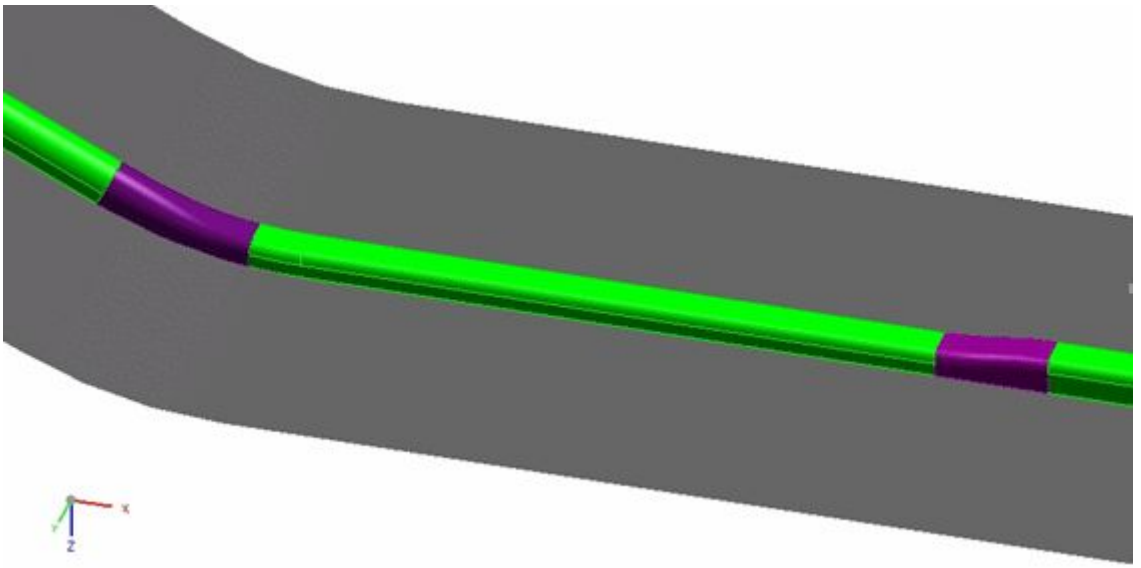
- The taper follows the centerline instead of a linear extension of the centerline.



Male draw bead with tapers

1. Spherical taper
2. Washout taper
3. Centerline

- You can adjust the blend lengths between two male beads.



Male bead with different blend lengths

Why should I use it?

Beads are useful design features in automotive body engineering. You can apply beads to sheet metal parts to strengthen the material. During sheet metal forming, beads directly influence the draw by controlling the flow of material in the die.

Where do I find it?

In the Modeling application, do one of the following:

- Choose **Tools® Vehicle Manufacturing Automation® Die Engineering® Draw Bead**.
- On the **Die Engineering** toolbar, click **Draw Bead** .

Die Validation

What is it?

With the enhanced Die Validation functionality, you can:

- Automatically mount Die Design features to the press model.

NX automatically mounts feature bodies created in Die Design to the press model used in Die Validation. NX uses the feature types, cam direction, and proximity of the base to choose the appropriate mount point in the press model. Automatic mounting occurs as soon as you define the Die Design data and a press model is available. You can turn off automatic mounting using a customer default, and let NX choose a mount point best suited to the data.

- Remove unnecessary updates from Die Validation functions.

  The performance of Die Validation functionality was impaired by repeated and excessive updates, which occurred even when no data changed. These updates are removed.

- Use faceted representation of bodies as input.

  Previous versions of Die Validation accepted only solid bodies or sheet bodies as input for analysis functions. The current version accepts faceted bodies. You can now load your assemblies with just the faceted representation of bodies displayed in the reference set, and select a faceted body for analysis. As NX needs a non-faceted body for analysis, the faceted bodies must have associated bodies.

  Mount, **Linear Cam**, and **Rotary Cam** functions will accept faceted bodies as input. **Run Simulation** is not affected by this change.

- Preview existing cams.

  You can select a cam from a list in the **Linear Cam** or **Rotary Cam** dialog boxes, and preview the motion associated with the cam.

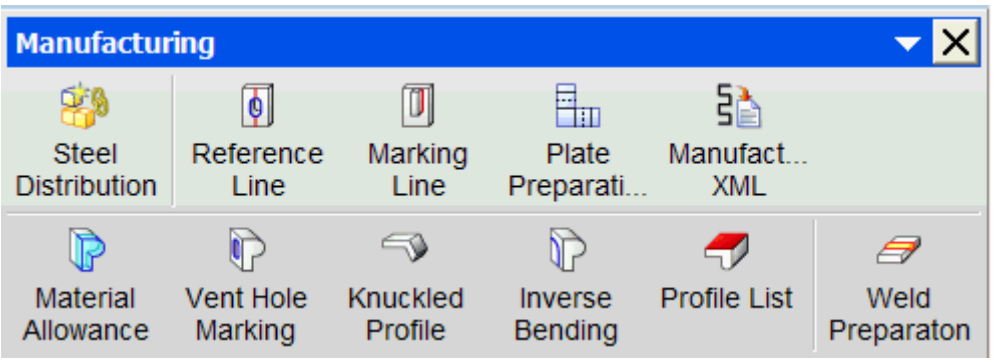
Where do I find it?

In the Modeling application:

- Choose **Tools**® **Vehicle Manufacturing Automation**® **Die Validation**.

Ship Design

Manufacturing features



Manufacturing toolbar

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 plane is a datum plane created by the **Concept** module.

Why should I use it?

You can 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 output 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 attributes 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?

You would 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 Profiles

What is it?

Knuckled Profiles 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 Profiles** provides this capability.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing® Knuckled Profiles**
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.

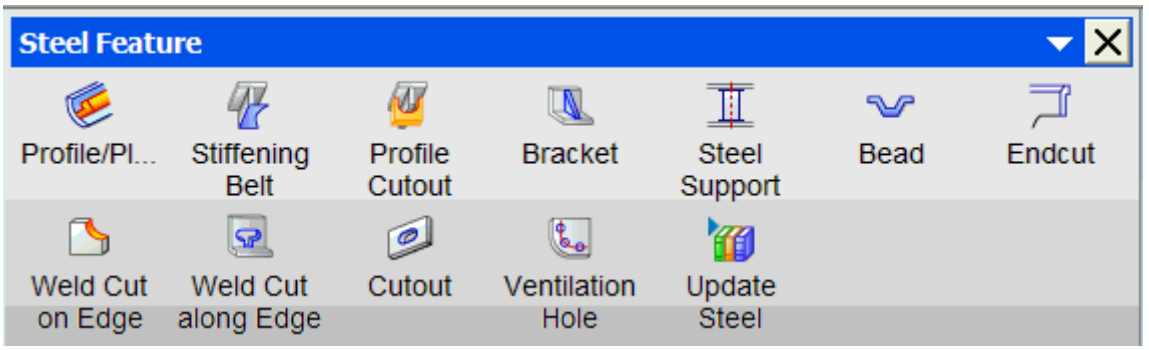
Why should I use it?

You would 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 it proper size and shape.

Where do I find it?

Application	Ship Design
Toolbar	**Manufacturing® Weld Preparation**
Menu	**Insert→Manufacturing→Weld Preparation**

Steel Features



Steel Feature toolbar

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**

Distribution

To access the four new manufacturing features in Ship Design you will need the menu and toolbar files. You can get these files by contacting Siemens PLM Software GTAC (Training & Support).

What is it?

During the process of designing ship parts, files are created containing multiple solid bodies. These solid bodies are steel features that represent the

steel structure of the ship. The **Distribution** function creates individual part files with a single steel feature or a section of the hull, deck, or bulkhead.

Why should I use it?

Use this function to separate the solid bodies contained in one file into separate files. Having a single solid body in a part file helps when manufacturing the parts.

Where do I find it?

In the Ship Design application:

- Choose **Insert→Manufacturing→Distribution**.
- On the **Manufacturing** toolbar, click **Distribution** .

Ship Flat Pattern

What is it?

Ship Flat Pattern flattens any ship bead or straight brake part that was separated into its own part file with the **Distribution** function.

Why should I use it?

Use this function to create a flat pattern representation of a ship bead or straight brake part. The flat pattern can then be used to create the part during the manufacturing process.

Where do I find it?

In the Ship Design application:

- Choose **Insert→Manufacturing→Ship Flat Pattern**.
- On the **Manufacturing** toolbar, click **Ship Flat Pattern** .

Reference Line

What is it?

Reference Line allows you to create an intersection curve between a grid datum plane offset from a Ship Grid Datum and a face.

Why should I use it?

Use this function when direct measurements from a datum plane representing a frame, deck, or bulkhead is not possible.

Where do I find it?

In the Ship Design application:

- Choose **Insert→Manufacturing→Reference Line**.
- On the **Manufacturing** toolbar, click **Reference Line** .

Profile List

What is it?

Use **Profile List** to produce a manufacturing list of all of the profiles in a distributed assembly, after you use **Distribution** to separate steel features into their own part files.

Where do I find it?

In the Ship Design application:

- Choose **Insert→Manufacturing→Profile List**.
- On the **Manufacturing** toolbar, click **Profile List** .

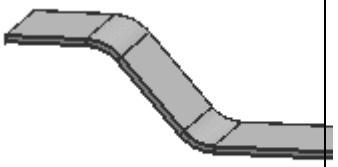
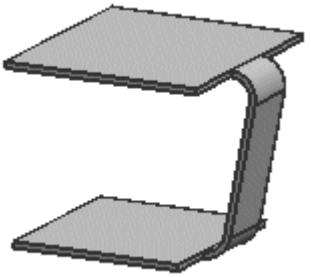
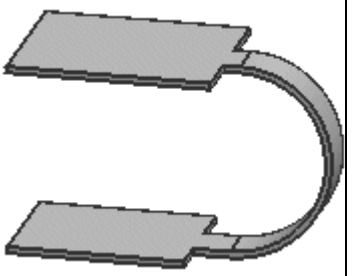
Flexible Printed Circuit Design

Bridge Transition

What is it?

Bridge Transition creates a shaped region that connects two distinct planar regions.

You can create a **Z**, **U**, or **Fold** shape, depending on the location and position of the geometry you want to connect.

Z	U	Fold
Consists of a planar region between two cylindrical bend regions. The axes of the bend regions are on opposite sides of the planar region. You can supply inner radii for the bends.	Consists of a planar region between two cylindrical bend regions. The axes of the bend regions are on the same side of the planar region. You can supply inner radii for the bends.	Consists of one cylindrical bend region between two planar regions.You supply a value to change the length of the planar region adjacent to the **Start Edge**.
		

Why should I use it?

This feature supports the typical workflow of flexible printed circuit design, which often begins with distinct planar regions and connects them with a transition region. This transition region can include bend as well as planar segments within it. This feature is an essential tool for designers of flexible printed circuits.

Where do I find it?

Choose **Insert→Flexible Printed Circuit Design Feature→Bridge Transition**.

On the **Flexible Printed Circuit Design** toolbar, click **Bridge Transition** .

PCBxchange

NX 5.0.2 Enhancements

Export multi-loop restriction areas individually

What is it?

Each loop in a multi-loop restriction area can now be exported to ECAD as a separate restriction area with identical attributes. You can modify the attributes in ECAD, or in PCB.xchange after importing the PCB assembly again.

Previously, a restriction area with multiple loops was exported to ECAD as a single restriction area. This is still the default.

To use the new functionality, you must modify the **NxSplitMultipleLoopAreas** variable in the *pcbx_ug_model.ini* file.

Why should I use it?

In NX Modeling, it is easier to create many restriction areas in a single sketch than to create a separate sketch for each restriction area. The new option allows you to define many restriction areas with a single sketch, yet process them as separate restriction areas in ECAD.

Where do I find it?

1. In a text editor, open the *pcbx_ug_model.ini* file in the *[NX INSTALLATION]/ugpcbxchange/* directory.

2. Modify the **NxSplitMultipleLoopAreas** variable value to **Yes**.

Solid and sheet bodies as restriction areas

What is it?

You can now define a restriction area by selecting a solid body or a sheet body.

Previously, you could select only sketches or faces for this purpose.

When you define a solid body as a restriction area, the height of the solid body represents the height of the restriction area.

Why should I use it?

Using a solid body or a sheet body rather than a sketch or a face makes it easier to visualize the design intent of a restriction area.

If you use solid bodies to define height restrictions, you can run an interference check to validate your design.

Where do I find it?

Define solid and sheet bodies as restriction areas as follows:

- On the **PCB.xchange toolbar**, select **Keep-in Area Attributes** or **Keep-out Area Attributes**, then select one or more solid or sheet bodies.

NX 5.0.3 Enhancements

Optional component designations on import or export

What is it?

For importing or exporting an ECAD model, you now have the option of modifying the default behavior for the translation of component designations. The following tables list the default translations and new optional translations.

	Designation on import to NX	
ECAD designation	**Default translation**	**New optional translation**
Component name (also called package, geometry or footprint name)	Part name	Part number (stored as component attribute in NX).
Component part number	Part number (stored as component attribute in NX)	Part name

	Designation on export to ECAD	
NX designation	**Default translation**	**New optional translation**
Part name	Component name (also called package, geometry or footprint name)	Component part number
Part number (stored as component attribute in NX)	Component part number	Component name (also called package, geometry or footprint name)

Where do I find it?

Importing using the new optional translation:

Application	PCB.xchange
Initialization file	*%MAYA_PCB_DIR%\pcbx_ug_model.ini* (in UNIX, *$MAYA_PCB_DIR/pcbx_ug_model.ini*)
Variable name	**NxWritePartNameNumSwap**
Variable value	**Yes**

Exporting using the new optional translation:

Application	PCB.xchange
Initialization file	*%MAYA_PCB_DIR%\pcbx_ug_model.ini* (in UNIX, *$MAYA_PCB_DIR/pcbx_ug_model.ini*)
Variable name	**NxReadPartNameNumSwap**
Variable value	**Yes**

Custom filter to specify reference sets

What is it?

When you import a PCB assembly from ECAD using MCAD components found in specified search folders or the Teamcenter database, you can now create a filter to specify which reference set to use for an individual MCAD component. You create the filter the same way you create any filter, and activate it by selecting it from the **Filters** List.

Why should I use it?

Components stored as MCAD part files in specified search folders or the Teamcenter database can include more than one configuration or version defined as a reference set. This filter makes it possible to use reference sets to extend the functionality of MCAD part collections with reference sets.

Where do I find it?

To create a filter that defines reference sets for individual components:

Application	PCB.xchange
Initialization file	*%MAYA_PCB_DIR%\pcbx_ug_filter.ini* (in UNIX, *$MAYA_PCB_DIR/pcbx_ug_filter.ini*)
Variable name	`[Filter Name]ComponentMapFile`
Variable value	[path and file name of text file]

Default assembly name for imported ECAD file

What is it?

When you import an ECAD file, a default name for the assembly part appears in the **Output Part** box in the dialog box. You can now choose one of three options to control how PCB.xchange creates this default name. You set the option by editing the initialization file *pcbx_ug_model.ini*.

In the initialization file, you can set one of the following variable options to control how PCB.xchange creates the default assembly part name:

- **None**. The box remains empty for you to type the name you want.
- **NX**. The name of the current NX part is displayed.
- **ECAD**. The name of the ECAD file you imported is displayed but the extension is changed to *.prt*.

You can also define a prefix or suffix to be added to the name, by editing the value of one or both the following variables in the initialization file *pcbx_ug_model.ini*:

- **NXWritePCANamePrefix**
- **NXWritePCANameSuffix**

Where do I find it?

The default assembly name for import:

Application	PCB.xchange
Initialization file	*%MAYA_PCB_DIR%\pcbx_ug_model.ini* (in UNIX, *$MAYA_PCB_DIR/pcbx_ug_model.ini*)

Variable name	`NXWritePCAName`
	`NXWritePCANamePrefix`
	`NXWritePCANameSuffix`

Variable value	[user defined text string]

Other Area restriction area

What is it?

You can create a new kind of restriction area, the Other Area restriction area. Its purpose and use is similar to the other two kinds of restriction areas, Keep-in Area and Keep-out Area. By default, the **Type** menu in the **Other Area** dialog box contains only one option, **Other**. The other options on the dialog box are identical to the **Keep-in Area** and **Keep-out Area** dialog boxes.

Why should I use it?

Use **Other Area** to define any area of interest on the PC board that is not a Keep-in Area or a Keep-out Area. If you need to create a special type of Other Area, you can do so by following the instructions for customized types for restriction areas

Where do I find it?

Other Area

Application	PCB.xchange
Toolbar	PCB.xchange® **Other Area**
Menu	PCB.xchange® **Other Area**

Customized types for restriction areas

What is it?

You can now define specialized types for keep-in areas, keep-out areas, and other areas. You:

- Define them by editing the initialization file *pcbx_ug.ini*.
- Can name a new type with any text string.
- The new type appears on the **Type** list in restriction area dialog boxes.

Where do I find it?

Application	PCB.xchange
Initialization file	*%MAYA_PCB_DIR%\pcbx_ug.ini* (in UNIX, *$MAYA_PCB_DIR/pcbx_ug.ini*)

Variable names	`KeepInAreaType`
	`KeepOutAreaType`
	`OtherAreaType`

Import Conductors as Curves

What is it?

ECAD conductors are now supported for import as follows:

- All conductors types are supported, including include traces, pads, filled areas and similar electrical features.
- Conductors are imported as curves.
- The curves are generated on the correct layer of the board part.

You can import conductors defined in ECAD files in IDF 4 format.

Where do I find it?

Application	PCB.xchange
Prerequisite	An ECAD model in IDF 4 file format, that includes conductors.
Toolbar	PCB.xchange® **Import ECAD Model**
Menu	PCB.xchange® **Import ECAD Model**

Digital Simulation

NX 5.0.1 Enhancements

Design Simulation

Centrifugal pressure loads

What is it?

Beginning with this release, Design Simulation now includes the new **Centrifugal Pressure** command. This new feature enables you to create and analyze radial varying centrifugal pressure loads, a feature similar in functionality to the existing **Hydrostatic Pressure** command. Centrifugal pressure loads occur whenever there is a liquid spinning in an object where the liquid is moving away from the axis of rotation.

Why should I use it?

If you need to model liquid spinning in an object, where the liquid is moving away from the axis of rotation, you can now use **Centrifugal Pressure** to calculate a centrifugal radial varying pressure load using the liquid density, angular velocity of the spinning vessel, a static pressure, and the radial offset (if not centered) from the location where the liquid is injected. Examples of radial varying centrifugal pressure loads include, but aren't limited to:

- Rotating pistons.
- Rotating drums, such as centrifuges, washing machines, and dental centrifugal casting machines.
- Rotating turbines, such as those found in hydroelectric power plants.

Where do I find it?

With the Simulation file active:

- On the **Design Simulation** toolbar, click **Centrifugal Pressure** .
- In the **Simulation Navigator**, right-click the **Load Container** and choose **New Load® Centrifugal Pressure**.
- In the **Simulation Navigator**, on the **Solution** node, right-click **Loads** and choose **New Load® Centrifugal Pressure**.
- Select geometry, then right-click **Create Load** and choose **Centrifugal Pressure**.

Expanded support of JT files for CAE data

What is it?

Beginning in this release, you can now export a much broader range of CAE data to JT (.jt) files. JT files provide you with a very lightweight representation of your model's data that you can then view in a supported viewer, such as Teamcenter Visualization. Exporting CAE data as JT files allows users, for example, to more easily share FE model data in a format that is both compact and easily transmitted throughout an organization.

In previous releases, when you were post-processing your model, you could create a JT file of selected results from your analysis once you had solved the model. In this release, you can now also create a JT file that contains a representation of all the FE entities in your model, including elements (element faces and edges), mesh mating conditions, loads, constraints, and solver-specific simulation objects, such as surface-to-surface contact.

With this capability, you can either:

- Manually export a JT file from the current FEM or Simulation file using the **Export** command on the **File** menu. With this option, the software includes all the entities which are currently visible in the graphics window. If the visibility of certain meshes or loads, for example, is turned off when you export the data to a JT file, the software does not include those meshes or loads in the JT file. Similarly, the software honors other current display settings, such as the **Element Shrink Percentage** option in the **Mesh Display** dialog box or whether internal element faces on solid meshes are currently displayed, when it creates the JT file.

- Automatically generate JT data from the current FEM or Simulation file each time you save using the **Save JT Data** option in the **Save Options** dialog box. With this option, the software outputs all supported data to the JT file each time you save your model, regardless of whether it is currently visible or not. With this option, the software does not honor current display settings, such as the **Element Shrink Percentage** option in the **Mesh Display** dialog box.

  When you use **Save JT Data**, the software creates the JT file in the same directory as the FEM or Simulation file to which it is associated. The software derives the name of the JT file from the name of the associated FEM or Simulation file.

  JT files are managed in Teamcenter Engineering in conjunction with Teamcenter Integration for NX. If you are in Teamcenter Integration mode using the Teamcenter for Simulation data model, the software creates a DirectModel dataset attached to the same ItemRevision where the FEM or Simulation file is stored. The generated JT file is stored as a named reference to this dataset.

Where do I find it?

- Choose **File**® **Export**® **JT**.
- Choose **File**® **Options**® **Save Options**.

Advanced Simulation

Teamcenter Integration for NX version support

What is it?

Teamcenter Integration for NX for NX now includes support for the following Teamcenter versions:

- Teamcenter 2007.1
- Teamcenter 2007.1 using the Teamcenter for Simulation data model

General capabilities

Better view synchronization between FEM and Simulation files

What is it?

The software now provides improved view synchronization when you switch between associated FEM (.fem) and Simulation (.sim) files. Because the FEM and Simulation files are separate files, each file contains its own separate view and display settings. In previous releases, there was no synchronization between those settings. For example, if you zoomed in very close on a region in your model in the FEM file, and then switched your active part to the Simulation file, your display changed to the view of the model when you had last displayed the Simulation file.

Beginning in this release, the software now synchronizes many view and display settings between the FEM and Simulation files. These settings include:

- View settings, such as the current zoom location and eye position.
- Display settings, such as whether the model displays in shaded or wireframe mode.
- The **Show/Hide** settings for the different objects in your model.

Display improvements for nodes and elements

What is it?

This release includes improvements to the display of nodes and elements.

- The software now always displays free nodes (nodes that are not associated with any elements) as asterisks, regardless of the current setting for the **Node Marker** option in the **Node and Element Display** dialog box. This makes it easy for you to locate available free nodes in your model without requiring that you display all the nodes in your model. This is helpful, for example, when you are using the new **Spider Element Create** capability to create connection elements, and you need to select a node to use as the element's core node. For more information on spider elements, see Enhanced support for connection elements.

- You can now use the following display commands to control the visibility of elements in your model:
  - **Show**
  - **Hide**
  - **Invert Shown and Hidden**
  - **Show All**
  - **Show Only**
  - **Show Adjacent**

  These commands give you more granular control over which elements are displayed. For example, you can use the **Hide** command to remove selected individual elements from your display.

- You can now use both the **Show by Name** and **Show All of Type** commands to display meshes in your model.

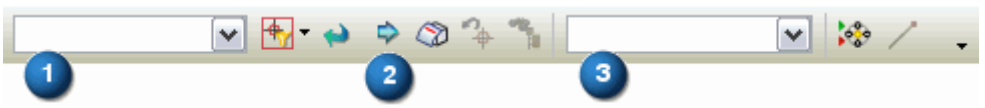
Where do I find it?

Choose **Edit→Show and Hide**, then choose the appropriate command.

New smart selection methods for CAE entities

What is it?

Additional smart selection methods have been added to the **Method** list on the **Selection Bar** to allow you to select CAE entities easily and more efficiently.



The Selection Bar, showing (1) the Type Filter list, (2) the Smart Selector Options button, and (3) the Method list

The new methods are **Feature Angle Nodes** and **Related Faces**.

- With the **Feature Angle Nodes** option, you choose **Feature Angle Nodes** from the **Method** list and then select an element face in the graphics window. The software then selects all the nodes on the element faces that are tangent to the initial element face. These are nodes that belong to the elements where the feature angle between the adjacent element face normal vectors is less than the **Feature Angle** value specified in the smart selector **Options** dialog box.

- With the **Related Faces** option, you first choose **Related Faces** from the **Method** list and then choose either **Node** or **Element** from the **Type Filter** list. Next, you select a node or element in the graphics window. The software then selects the face or faces associated with that node or element.

Expanded support of JT files for CAE data

What is it?

Beginning in this release, you can now export a much broader range of CAE data to JT (.jt) files. JT files provide you with a very lightweight representation of your model's data that you can then view in a supported viewer, such as Teamcenter Visualization. Exporting CAE data as JT files allows users, for example, to more easily share FE model data in a format that is both compact and easily transmitted throughout an organization.

In previous releases, when you were post-processing your model, you could create a JT file of selected results from your analysis once you had solved the model. In this release, you can now also create a JT file that contains a representation of all the FE entities in your model, including elements (element faces and edges), mesh mating conditions, loads, constraints, and solver-specific simulation objects, such as surface-to-surface contact.

With this capability, you can either:

- Manually export a JT file from the current FEM or Simulation file using the **Export** command on the **File** menu. With this option, the software includes all the entities which are currently visible in the graphics window. If the visibility of certain meshes or loads, for example, is turned off when you export the data to a JT file, the software does not include those meshes or loads in the JT file. Similarly, the software honors other current display settings, such as the **Element Shrink Percentage** option in the **Mesh Display** dialog box or whether internal element faces on solid meshes are currently displayed, when it creates the JT file.

- Automatically generate JT data from the current FEM or Simulation file each time you save using the **Save JT Data** option in the **Save Options** dialog box. With this option, the software outputs all supported data to

the JT file each time you save your model, regardless of whether it is currently visible or not. With this option, the software does not honor current display settings, such as the **Element Shrink Percentage** option in the **Mesh Display** dialog box.

When you use **Save JT Data**, the software creates the JT file in the same directory as the FEM or Simulation file to which it is associated. The software derives the name of the JT file from the name of the associated FEM or Simulation file.

JT files are managed in Teamcenter Engineering in conjunction with Teamcenter Integration for NX. If you are in Teamcenter Integration mode using the Teamcenter for Simulation data model, the software creates a DirectModel dataset attached to the same ItemRevision where the FEM or Simulation file is stored. The generated JT file is stored as a named reference to this dataset.

Where do I find it?

- Choose **File**® **Export**® **JT**.
- Choose **File**® **Options**® **Save Options**.

Meshing

Enhanced support for connection elements

Note

In NX 6, the **Spider Element Create** command has been replaced by the **1D Connection** command. For more information, see 1D Connection.

What is it?

Beginning with this release, you can use the **Spider Element Create** dialog box to create a rigid or constraint type element (depending on the solver) that connects a single node (the core node) to multiple nodes (leg nodes).

When defining a spider element, the first node you select becomes the core node and all subsequent selections become leg nodes. A group of smart selection methods is available to help you select the leg nodes.

Note

For more information about the new smart selection methods, see New smart selection methods for CAE entities.

After creating the spider element, you can make specific degrees of freedom active or inactive by editing the element attributes.

Supported element types depend on the solver you use.

Solver	Spider Element type
NX Nastran/MSC Nastran	RBE2
	RBE3
ANSYS	CERIG
ABAQUS	Kinematic Coupling

For information about the element types, see your solver documentation.

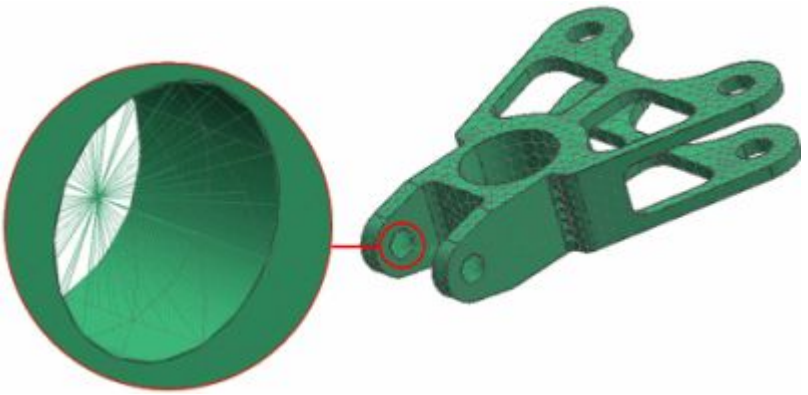
Why should I use it?

In previous releases, in Nastran models, the software generated a separate RBE2 element for each leg node in the Nastran input file. Also, the display of the spider mesh was cluttered with an RL symbol for each leg node. RBE3 elements were not supported.

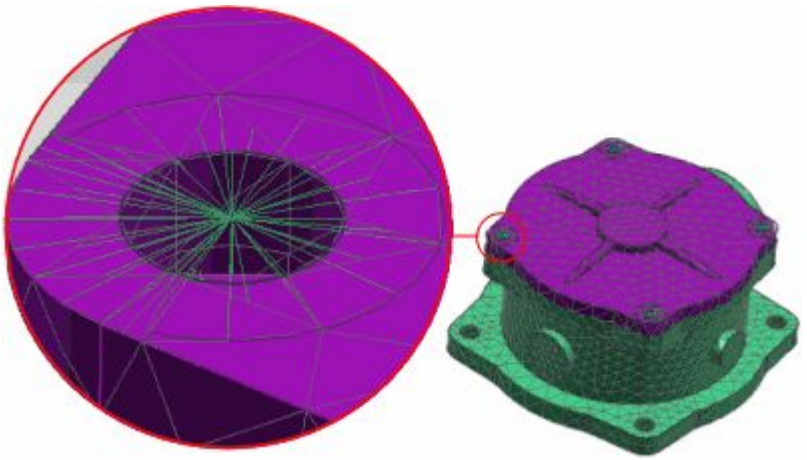
In this release, the creation of the mesh is more automated, creates only a single RBE2 or RBE3 bulk data entry in the Nastran input file for each spider element, and displays the element more neatly in the graphics window.

Typical uses for spider elements include:

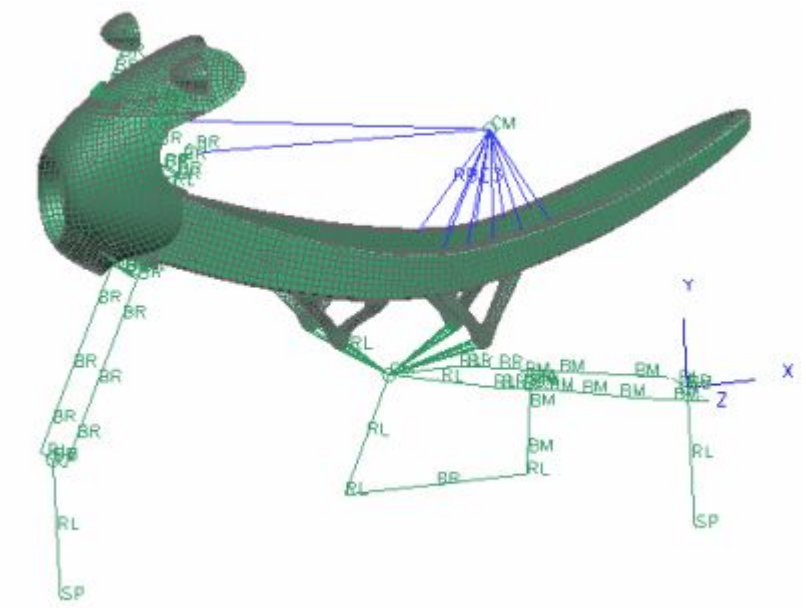
- To represent a pin in a hole. In the example below, the pin is modeled using two spider elements and a beam element. The core node of the spider element is defined at the center of the hole, and the leg nodes connect to the mesh on the inner face of the hole.



- To represent a bolt. In the example below, the head of the bolt is modeled with a spider element and the shank is modeled with a beam element. After modeling the bolt, you can apply a pretension load using the Bolt Pre-Load boundary condition. For information about Bolt Pre-Load, see Bolt pre-loads.



- To add and distribute a mass or a load. In the motorcycle example below, an RBE3 spider element distributes the rider's mass (represented by the concentrated mass element) to the seat and handlebars. The RBE3 is used in this case because it adds the mass without adding stiffness.



Where do I find it?

With the FEM file displayed:

- On the **Element Operations** toolbar, click **Spider Element Create** .
- Choose **Edit® Element® Create Spider**.

Element support improvements

What is it?

In this release, support has been added for a number of new element types for the NX Nastran and MSC Nastran solvers. Additionally, this release also includes improved support for several NX Nastran, MSC Nastran, and ANSYS element types available in previous releases.

Newly supported NX Nastran and MSC Nastran element types

Element Name	Description
CBUSH, node-to-ground	Generalized spring and damper connection
CDAMP1, node-to-ground and node-to-node	Scalar damper connection
CDAMP2, node-to-ground and node-to-node	Scalar damper element (without reference to a property entry)
CELAS1, node-to-ground and node-to-node	Scalar spring element
CELAS2, node-to-ground	Scalar spring element (without reference to a property entry)
CGAP	Gap or friction element
CMASS1, node-to-ground and node-to-node	Scalar mass element
CMASS2, node-to-ground and node-to-node	Scalar mass element (without reference to a property entry)
CONM1	Concentrated mass element
CONROD	Rod element (without reference to a property entry)
CSHEAR*	Shear panel element
CTUBE	Tube (tension-compression-torsion) element
CVISC	Viscous damper element
PLOTEL	Dummy 1D element for use in plotting
RBAR	Rigid bar element
RBE3	Interpolation constraint element
RROD	Rigid pin-ended connection element

*The CSHEAR element is only available from the **2D Mapped Mesh** dialog box. Because the CSHEAR element has four nodes, and there is no related three-noded transition element for a CSHEAR element, you can only use it in the context of a structured mesh.

Enhancements to previously supported NX Nastran and MSC Nastran elements

Element Name	Description	Enhancements

CBAR	Beam element	You can now define pin flags to specify beam end releases.
CBEAM	Beam element	You can now define pin flags to specify beam end releases.
CBUSH, node-to-node	Generalized spring and damper connection	You can now define the location and offset for the spring-damper of this element.
CELAS2, node-to-node	Scalar spring element	You can now specify a stress coefficient for this element.
RBE2	Rigid body element, form 2	You can now create RBE2 elements that have more than one dependent node. See Enhanced support for connection elements for more information.

Enhancement to previously supported ANSYS element

Element Name	Description	Enhancements
PRETS179	Pretension element	This element is now supported in axisymmetric structural type analyses.

Where do I find it?

With the FEM file displayed:

- On the **Advanced Simulation** toolbar, click **0D Mesh** ,**1D Mesh** , or **2D Mapped Mesh** .
- Choose **Insert→Mesh→0D Mesh**, **1D Mesh**, or **2D Mapped Mesh**.

Limited support for pyramid elements at hexahedral-tetrahedral mesh interfaces

What is it?

Note

In NX6, the pyramid element capabilities have been expanded to support transitions between meshes of different orders. See Expanded pyramid element support for more information.

This release contains the initial implementation of pyramid element support. In Advanced Simulation, if you are working in the ANSYS solver language, you can now use pyramid elements to transition between hexahedral and tetrahedral meshes on adjacent bodies (volumes). Pyramid elements help ensure a conforming mesh as they provide a more direct transition from

hexahedral elements to tetrahedral elements than interface methods that rely on either rigid elements or multi-point constraint equations to connect the nodes. Using pyramid elements to join dissimilar meshes also offers advantages in greater solution accuracy and reduced solution time over rigid elements or multi-point constraint equations.

Beginning in this release, you can now:

- Create pyramid elements at the interface between hexahedral and tetrahedral meshes.
- Import existing ANSYS models that contain pyramid elements and work with those elements in Advanced Simulation.

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 new **Use Pyramids for Transition** option on 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.
- Currently, the order of the tetrahedral elements and hexahedral elements must match. In other words, you can use pyramid elements to transition from a mesh of linear hexahedral elements to a mesh of linear tetrahedral

elements or from a mesh of parabolic hexahedral elements to a mesh of parabolic hexahedral elements. However, in this release, you cannot use pyramid elements to transition from a mesh of linear hexahedral elements to a mesh of parabolic tetrahedral elements, and vice versa.

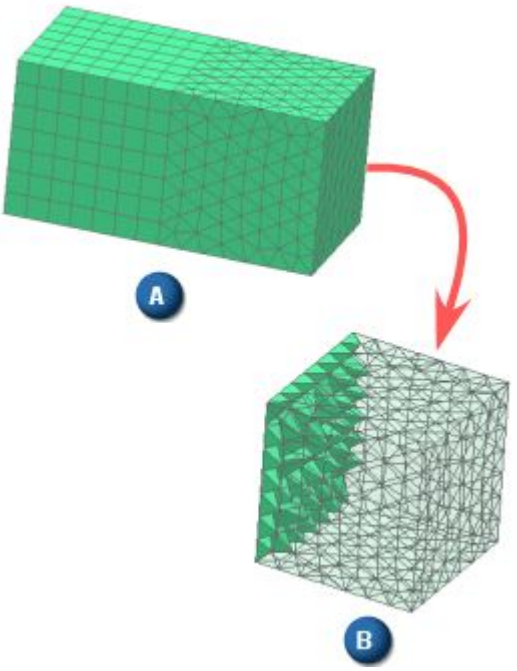
- A **Glue Coincident** type mesh mating condition must be defined at the interface between the two bodies to ensure that the nodes on the adjacent hexahedral and tetrahedral meshes match exactly.

In ANSYS, pyramid elements are simply degenerate forms of certain types of linear and parabolic solid elements. The type of pyramid element the software creates depends upon whether you are connecting linear or parabolic hexahedral and tetrahedral elements.

- If you connect linear hexahedral to linear tetrahedral elements, the software creates 5-noded pyramid elements.

- If you connect parabolic hexahedral to parabolic tetrahedral elements (Hex20 to Tet10), the software creates 13-noded pyramid elements.

The following example shows the use of transitional pyramid elements at the interface of hexahedral and tetrahedral meshes on a simple, partitioned block. (A) shows a side view of the interface between the two different meshes, while (B) shows a view of just the pyramid elements with the tetrahedral elements.



Importing ANSYS models that contain pyramid elements

Beginning in this release, you can now import ANSYS models that contain pyramid elements into Advanced Simulation. In previous releases, if you

imported an ANSYS model that contained pyramid elements, you had no way to display or manipulate those pyramid elements in Advanced Simulation.

Note

Although ANSYS allows you to create pyramid elements in which certain midside nodes are removed, Advanced Simulation does not currently support that capability. If you import an ANSYS model that contains pyramid elements with removed midside nodes, Advanced Simulation skips those elements during the import operation.

Where do I find it?

With the FEM file displayed:

- On the **Advanced Simulation** toolbar, click **3D Mesh** .
- Choose **Insert→Mesh→3D Mesh**.

Loads and boundary conditions

Centrifugal pressure loads

What is it?

Beginning with this release, Advanced Simulation now includes the new **Centrifugal Pressure** command. This new feature enables you to create and analyze radial varying centrifugal pressure loads, a feature similar in functionality to the existing **Hydrostatic Pressure** command. Centrifugal pressure loads occur whenever there is a liquid spinning in an object where the liquid is moving away from the axis of rotation.

Why should I use it?

If you need to model liquid spinning in an object, where the liquid is moving away from the axis of rotation, you can now use **Centrifugal Pressure** to calculate a centrifugal radial varying pressure load using the liquid density, angular velocity of the spinning vessel, a static pressure, and the radial offset (if not centered) from the location where the liquid is injected. Examples of radial varying centrifugal pressure loads include, but aren't limited to:

- Rotating pistons.
- Rotating drums, such as centrifuges, washing machines, and dental centrifugal casting machines.
- Rotating turbines, such as those found in hydroelectric power plants.

Where do I find it?

With the Simulation file active:

- On the **Advanced Simulation** toolbar, click **Centrifugal Pressure** .
- In the **Simulation Navigator**, right-click the **Load Container** and choose **New Load® Centrifugal Pressure**.
- In the **Simulation Navigator**, on the **Solution** node, right-click **Loads** and choose **New Load® Centrifugal Pressure**.
- Select geometry, then right-click **Create Load** and choose **Centrifugal Pressure**.

Bolt pre-loads

What is it?

Beginning with the 5.0.1 release, NX now supports bolt pre-loads within Advanced Simulation. This capability allows you to model pre-loads as appropriate for the following solvers:

- ABAQUS
- ANSYS
- NX Nastran

Pre-loads in bolts often have significant effects on deflections and stresses. Whenever you need to model tightening forces or length adjustments in bolts or fasteners, use the new **Bolt Pre-Load** feature.

Why should I use it?

You may be interested in understanding the contact conditions with an applied service load after the pre-load, or in calculating the stresses from the combination of the pre-load and the service load. While you can perform such an analysis manually by using equivalent thermal loads, such methods are approximate and may require several iterations when multiple bolts are present. The **Bolt Pre-Load** feature provides you with a more automated, and more accurate, method for calculating the effects of these loads.

Bolt pre-load process

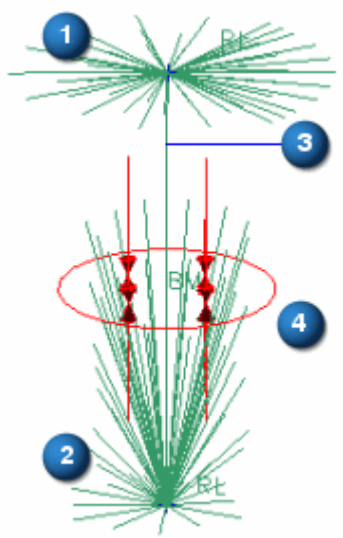
Regardless of the solver you use, the process for defining bolt pre-loads is basically the same.

1. With the FEM file active, model the head and nut using spider elements to connect the bolts into the mesh on the surrounding part. See Enhanced support for connection elements for more information.

2. Model the shank of the bolt as one or more elements, as required. (The appropriate type of element varies, depending upon your solver).

3. With a Simulation file active, define the pre-load boundary condition for the bolt. (You can define a single pre-load on multiple bolts at the same time.)

4. Solve the model.



In the illustration above:

1 The spider element representing the head of the bolt.

2 The spider element representing the end of the bolt. (In this case, it represents a bolt threaded into a tapped hole; if it were on the bottom edge or face of the model, a slightly different spider element would represent a through-bolt with a nut on the other end).

3 The 1D element (in this case a CBEAM) representing the bolt's shank.

4 The graphical representation of the applied bolt pre-load (in orange).

Where do I find it?

With a Simulation file active:

- On the **Advanced Simulation** toolbar, click **Bolt Pre-Load** .

- In the **Simulation Navigator**, right-click **Load Container** and choose **New Load® Bolt Pre-Load**.

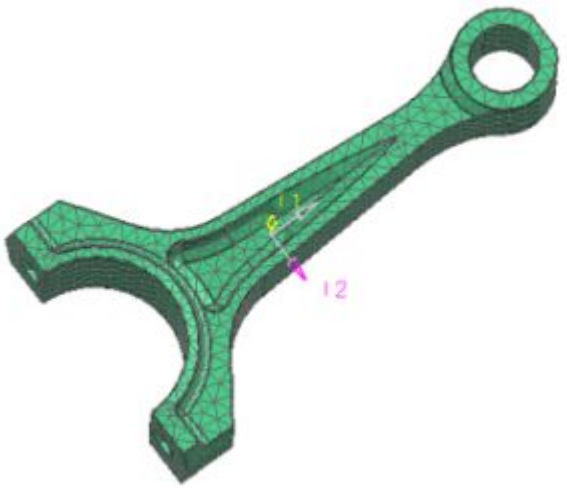
- In the **Simulation Navigator** under the **Solution** node, right-click **Loads** and choose **New Load® Bolt Pre-Load**.

New model validation tools

What is it?

This release includes two new commands that help you verify aspects of your model prior to performing a solve.

- The **Solid Properties** command lets you verify the solid properties of the elements in your model. You can use **Solid Properties** to calculate the total mass of your model, including structural and nonstructural mass, along with the principal moments of inertia. You can also use **Solid Properties** to have the software graphically indicate the location of the model's center of gravity along with the principal axes about the center of gravity. **Solid Properties** is useful, for example, if you want to validate that the mass of the FEM appropriately matches the mass of the associated part.



  If you are working in the NX Thermal and Flow, NX Electronic Systems Cooling, or NX Space Systems Thermal environment, the **Solid Properties Check** also calculates the surface area for convection and radiation as well as the thermal capacitance of your model.

- The **Mechanical Load Summary** command lets you verify that you have correctly applied structural loads in your FEM file. With **Mechanical Load Summary** you can compute the total forces and moments for selected solutions or subcases in your model. You can use **Mechanical Load Summary** to verify both geometry-based and FE-based loads within structural analysis type solutions.

Where do I find it?

Solid Properties:

- With the FEM or Simulation file displayed, choose **Information→Advanced Simulation→Solid Properties Check**.

Mechanical Load Summary:

- With the Simulation file displayed, choose **Information→Advanced Simulation→Mechanical Load Summary**.
- From the Simulation Navigator, right-click the active solution or subcase and choose **Mechanical Load Summary**.

Solvers and solutions

Supported solver versions

What is it?

In this release, the software supports the following solver versions:

- NX Nastran 5 and earlier versions
- MSC Nastran 2005 and earlier versions
- ANSYS 10.0 and earlier versions
- ABAQUS 6.6 and earlier versions

Option to import forces or moments as field table

What is it?

The new **Import Force and Moment BCs as a Field Table** option in the Simulation **Customer Defaults** lets you automatically import Nastran, ANSYS, or ABAQUS forces or moments as a field table.

When you select this option, the software collates all force type boundary conditions that share a common coordinate system and displays them as a single field table in the **Simulation Navigator**. The software also collates all moment type boundary conditions that share a common coordinate system and displays them as a single field table. Previously, each imported boundary condition was displayed separately. This remains the default behavior.

Why should I use it?

When importing models with many force or moment boundary conditions, the new option improves performance and usability. The default method may

result in a slow import processing and require an unreasonable amount of scrolling to view the boundary conditions in the **Simulation Navigator**.

Where do I find it?

In Advanced Simulation:

1. Choose **File**® **Utilities**® **Customer Defaults**.
2. In the **Customer Defaults** dialog box, select **Simulation** from the list tree, and then select the **General** node.
3. On the Environment page of the **Customer Defaults** dialog box, select the **Import Force and Moment BCs as a Field Table** check box.

Nastran support enhancements

What is it?

This release includes expanded support for the Nastran solvers. For information on supported fields for specific Nastran case control commands and bulk data entries, see Nastran import and export support.

Surface-to-Surface Contact support extended

You can now include **Surface-to-Surface Contact** definitions in Nastran SOL 103, 111, and 112 analyses. You can also include **Surface-to-Surface Contact** definitions in NX Nastran SOL 103* (Response Simulation) analyses.

Surface-to-Surface Gluing support extended

You can now include **Surface-to-Surface Gluing** definitions in Nastran SOL 153 analyses and NX Nastran SOL 601, 106 and SOL 601, 129 analyses. In previous releases, you could not create **Surface-to-Surface Gluing** definitions in thermal or nonlinear analyses.

Rigid element support improvements

- In this release, you can now create RBE2 elements with more than one dependent node. This allows you to use RBE2 elements to create spider-type elements where a single independent node is connected to multiple dependent nodes. See Enhanced support for connection elements for more information.
- In Nastran analyses, the new **Rigid Element Method** option in the **Create Solution** dialog box lets you use the RIGID case control command to control how the software handles the specified thermal expansion coefficient of rigid elements. The element types affected by this option vary depending upon your solver:

– In NX Nastran, this option affects RBAR, RBE2, and RROD elements.

– In MSC Nastran, this option affects RBAR, RBE2, RBE3, and RROD elements.

In Advanced Simulation, you use the **Thermal Expansion Coefficient** option in the **Edit Attributes** dialog box to specify the coefficient. This corresponds to the ALPHA field on the appropriate rigid element bulk data entries in Nastran.

In Advanced Simulation, **Rigid Element Method** option is available in SOLs 101 through 112 for NX Nastran and SOLs 101, 103, 105, and 106 for MSC Nastran.

For the **Rigid Element Method** option in the **Create Solution** dialog box:

– Choose **Linear Elimination** to have the software treat rigid elements as multi-point constraint equations without thermal loading effects

– Choose **Lagrange Multiplier** to have the software include rigid elements in thermal expansion calculations.

If you are working with MSC Nastran, you can also choose the **Lagrange Multiplier with Elimination** (LGELIM) option. For more information on this option, see the Rigid case control entry in the *MSC Nastran Quick Reference Guide*.

Element support enhancements

This release includes added support for a number of new types of Nastran elements as well as extensions to elements supported in previous releases. See Element support improvements for more information.

Importing and exporting model data

Nastran import and export support

What is it?

This release includes import and export support for a number of new Nastran bulk data entries, as well as import and export support for additional fields on several bulk data entries that were supported in previous releases.

Newly supported bulk data entries

The table below lists the newly supported bulk data entries. Except as otherwise noted, these bulk data entries are:

- Supported for both NX Nastran and MSC Nastran.
- Supported for both ASCII or binary files.

In the table below, newly supported cards are listed in the left hand column. All fields are supported for these cards, except as otherwise noted in the right hand column.

New card	Unsupported fields	Notes
BGPARM		Supported for NX Nastran only.
BOLT		• Supported for NX Nastran only. • Not supported for binary (OP2) files.
BOLTFOR		• Supported for NX Nastran only. • Not supported for binary (op2) files.
CBUSH node-to-ground		• C1 and C2 can only be a single digit.
CDAMP1 node-to-ground		• Scalar points are imported as GRID points. • Scalar points must be defined using the SPOINT entry. • C1 and C2 can only be a single digit.
CDAMP1 node-to-node		• Scalar points are imported as GRID points. • Scalar points must be defined using the SPOINT entry.

New card	Unsupported fields	Notes
		• C1 and C2 can only be a single digit.
CDAMP2 node-to-ground		• Scalar points are imported as GRID points.
		• Scalar points must be defined using the SPOINT entry. • C1 and C2 can only be a single digit.
CDAMP2 node-to-node		• Scalar points are imported as GRID points.
		• Scalar points must be defined using the SPOINT entry.
CELAS1 node-to-ground		• Scalar points are imported as GRID points.
		• Scalar points must be defined using the SPOINT entry.
CELAS2 node-to-ground		• Scalar points are imported as GRID points.
		• Scalar points must be defined using the SPOINT entry.
CGAP node-to-node		
		• Scalar points are imported as GRID points.
CMASS1		
		• Scalar points must be defined using the SPOINT entry.

New card	Unsupported fields	Notes
CMASS2		• Scalar points are imported as GRID points. • Scalar points must be defined using the SPOINT entry.
CONM1	M41, M42, M43, M51, M52, M53, M61, M62, M63	
CONROD		
CSHEAR		
CTUBE		
CVISC		
PDMAP		
PGAP		
PLOTEL		
PMASS		
PSHEAR		
PTUBE		
PVISC		
RBAR		
RBE2		
RBE3	The UM fields are unsupported.	• All WT fields are set to 1.0. • All C fields have the same DOFs.
RROD		

Additional supported fields for previously supported bulk data entries

Card	Fields newly supported	Notes
CBUSH node to node	S, S1, S2, S3, and OCID	All fields are now supported.
CBEAM	PA and PB	BIT, SA, and SB fields remain unsupported.
CELAS2 node to node	S	Scalar points are imported as GRID points. Scalar points must be defined using the SPOINT entry.

Where do I find it?

To import a Nastran input file, in the Advanced Simulation and Gateway applications:

- Choose **File® Import® Simulation**.

To export a Nastran Simulation or FEM file:

- Choose **File® Export® Simulation**.

ABAQUS and ANSYS import and export support

What is it?

This release includes several improvements to the import and export support for the ABAQUS and ANSYS solvers.

ABAQUS improvements

- The following new keywords for defining thermal loads in ABAQUS are now supported on import: *CFLUX, *DFLUX, *CFILM, *DFILM, and *FILM.

- You can export the new **Bolt Pre-load** type boundary conditions from ABAQUS solutions. Currently, only the PRETENSION SECTION and *SURFACE keywords are supported. For more information on the new **Bolt Pre-load** boundary condition, see Bolt pre-loads.

ANSYS improvements

- You can import an ANSYS bolt pre-load boundary condition (SLOAD command) into Advanced Simulation. All fields are fully supported. For more information on the new **Bolt Pre-load** boundary condition, see Bolt pre-loads.

- You can export the new **Bolt Pre-load** type boundary conditions from ANSYS solutions. All fields on the SLOAD command are fully supported.

Where do I find it?

To import or export a file, from the Advanced Simulation or Gateway applications:

- Choose **File® Import® Simulation**.

- Choose **File® Export® Simulation**.

Import Simulation workflow improvements

What is it?

The **Import Simulation** workflow has been improved to provide better support for Teamcenter Integration for NX.

In the Import Simulation dialog box, specify the **Input File Units**, **File Type**, and **Input File**. When you click **OK**, a second dialog box appears:

- If you are running NX in native mode, you are prompted for the FEM file name, the Simulation file name, and the directory location.
- If you are running NX in Teamcenter Integration for NX mode using the traditional data model, you are prompted for the FEM name, the Simulation name, and the storage folder.
- If you are running NX in Teamcenter Integration mode using the Teamcenter for Simulation data model, you are prompted for the number, name, and revision for the FEM and the Simulation, and for the storage folder.

Where do I find it?

In the Advanced Simulation and Gateway applications, do the following in a part file, a FEM file, or a Simulation:

- Choose **File**® **Import**® **Simulation**.

NX Thermal and Flow

Thermal mapping target set

What is it?

The new **Mapping Target Set** constraint allows you to restrict the mapping algorithm to map either temperatures or fluid forces to selected nodes or elements in the target set. This constraint replaces the **Flow Mapping Target Set** constraint, which only controlled the mapping of fluid forces.

The new constraint maps fluid forces in exactly the same way as the previous constraint.

Why should I use it?

Previously, temperatures were mapped based only on the location of the target elements in relation to the source elements. The **Mapping Target Set** constraint allows you more control over which elements the mapping algorithm can consider.

Where do I find it?

In Advanced Simulation, in a **Mapping** solution, follow these steps in the target model to set up mapping temperatures to specific nodes:

1. In the **Simulation Navigator**, right-click the mapping solution and choose **Solution Attributes**.

2. In the **Edit Solution** dialog box, in the **Data to Map** group, choose **Solid Temperatures** from the **Fields** list.

3. In the **Simulation Navigator**, right-click the **Constraint Set** and choose **New Constraint® Mapping Target Set**.

4. In the **Mapping Target Set** dialog box, select the **Destination Model Elements** or **Destination Model Nodes**.

Turbulent data flow results

What is it?

Turbulent data is a results option that generates post processing data evaluating turbulence at nodes for each fluid element.

> **Note**
>
> These results are only generated when either the **K-epsilon** option or the **Shear Stress Transport** option is selected from the **Viscous Model** list in the **Edit Solution** dialog box.

Slightly different results are generated, depending on the viscous model for the solution.

- The **K-epsilon** option generates Turbulence Energy data and Turbulence Dissipation data.

- The **Shear Stress Transport** option generates Turbulence Energy data and Specific Dissipation Rate data.

Turbulence Energy results characterize the amount of turbulence. The two dissipation results types (Turbulence Dissipation and Specific Dissipation Rate) characterize the lack of turbulence. Note that the two values indicate the opposite characteristics.

Why should I use it?

Use these results types to gain an understanding of the turbulence distribution in the flow field.

Where do I find it?

In Advanced Simulation, do the following to set up the solution to generate turbulent data results:

1. In the **Simulation Navigator**, right-click the active solution and choose **Edit Solution**.

2. In the **Edit Solution** dialog box, click the **Results Options** tab and select the **Turbulent data** check box.

Separate convergence plots for multiple scalars

What is it?

If your flow model includes more than one scalar component, the solution monitor now displays a separate convergence plot for each scalar component.

Why should I use it?

Separate convergence plots can be essential to troubleshooting a model which is oscillating or diverging due to problems with scalar convergence.

Where do I find it?

In Advanced Simulation, with an analysis running or paused:

- In the **Solution Monitor** dialog box, click **Flow** to view all convergence plots.

Extract additional results

What is it?

A new button allows you to load additional results types for post-processing after the analysis is complete. Previously, you had to select all desired results types before running the analysis. To extract additional results after the analysis is complete, select the results type(s), and then click **Refresh** at the top of the Results Options page of the **Edit Solution** dialog box. This launches the results recovery module of the solver, and quickly recreates the results (.bun) file for post processing.

Why should I use it?

This feature makes it possible to first load only data of major interest into the results file. After post-processing these results, you may decide to discard your results and modify your model, or you may decide to extract additional results types from the same analysis. On the other hand, you may discover that you need additional results not foreseen when you selected results options on the **Edit Solution** dialog box before the analysis.

You avoid unnecessary processing time by loading only the results you know you need, yet you have quick access to all results without having to solving the model again.

Where do I find it?

In a simulation, with the solver type set to **NX Thermal / Flow**, do the following to extract additional results after the analysis is complete:

1. In the **Post-Processing Navigator**, right-click the **Results** node and select **Close** to unload previous results.

2. In the **Simulation Navigator**, right-click the active solution and choose **Edit Solution**.

3. In the **Edit Solution** dialog box, on the Results Options page, select or clear any check box to add additional results, or remove any previously extracted results, and then click **Refresh** at the top of the page to regenerate the results.

4. In the **Simulation Navigator**, double-click the **Results** node to see the new results sets.

Flow velocity gamma report

What is it?

Velocity Gamma Values is a new option for the **Per Region** report type that provides scalar data equal to the velocity gamma at nodes for the selected elements. Velocity gamma is an index that measures flow uniformity. A gamma value of unity indicates that the flow is totally uniform.

Why should I use it?

These results may be useful in determining whether or not a flow is fully developed, or to investigate flow uniformity in a specific part of the model.

Where do I find it?

In a solution, with the **Analysis Type** set to **Flow** or **Coupled Thermal-Flow**, do one of the following to create a **Per Region** type **Report**:

- In the **Simulation Navigator**, right-click **Simulation Objects** and choose **New Simulation Object® Report**. In the **Report** dialog box, select **Per Region**.

- On the **Advanced Simulation** toolbar, click **Simulation Object Type** **® Report** . In the **Report** dialog box, select **Per Region**.

Report results in spreadsheet format

What is it?

Some reported results specified in the **Report** simulation objects can now be generated as a comma separated value (.csv) file that can be opened in a spreadsheet program such as Microsoft Excel. The solver saves the file, groupReport.csv, in the run directory for the solution.

The results data stored in the file groupReport.csv are identical to the results data stored in the file groupReport.htm. See the NX 5.0 help for a full description of these results. Depending on the report type you create, the file contains the following data:

- **Between Regions —** Heat flow, view factors, radKs, and 3D flow pressure drop between selected regions of the model.

- **Per Region —** Temperature, heat load, heat flux, physical property, orbital and source view factors, phase change quality, and duct flow data.

- **Heat Map —** Heat load and heat flow into the selected elements via conduction, radiation, convection couplings, and thermal couplings. Create two or more objects, and the Heat Map report generates heat flow data between each pair of **Report** objects.

- **Lift and Drag —** Lift and drag forces on selected surfaces. Typically, lift and drag vectors do not correspond to the model's global coordinate system, making it difficult to study these forces. You can generate a report detailing lift and drag forces at the center of gravity of each element in the selected region, isolating these vectors for flight applications.

Why should I use it?

Generating this data in tabular text format makes it easier to use other software such as a spreadsheet program for additional data processing not available in NX Advanced Simulation post-processing.

Where do I find it?

In a solution, with the solver type set to **NX Thermal / Flow**, do the following to generate the file groupReport.csv in the run directory:

1. Do one of the following to open the **Report** dialog box:

    - In the **Simulation Navigator**, right-click **Simulation Objects** and choose **New Simulation Object® Report**. In the **Report** dialog box, select **Per Region**.

- On the **Advanced Simulation** toolbar, click **Simulation Object Type** ® **Report** . In the **Report** dialog box, select **Per Region**.

2. In the **Report** dialog box, select **Between Regions**, **Per Region**, **Heat Map**, or **Lift and Drag** from the **Type** list.

3. Solve the model.

Solid properties check

What is it?

In an NX Thermal and Flow analysis, you can use the new **Solid Properties Check** command to calculate the surface area for convection and radiation as well as the thermal capacitance of your model. See New model validation tools for more information.

NX Electronic Systems Cooling

Thermal mapping target set

What is it?

The new **Mapping Target Set** constraint allows you to restrict the mapping algorithm to map either temperatures or fluid forces to selected nodes or elements in the target set. This constraint replaces the **Flow Mapping Target Set** constraint, which only controlled the mapping of fluid forces.

The new constraint maps fluid forces in exactly the same way as the previous constraint.

Why should I use it?

Previously, temperatures were mapped based only on the location of the target elements in relation to the source elements. The **Mapping Target Set** constraint allows you more control over which elements the mapping algorithm can consider.

Where do I find it?

In Advanced Simulation, in a **Thermal-Flow Mapping** solution, follow these steps in the target model to set up mapping temperatures to specific nodes:

1. In the **Simulation Navigator**, right-click the mapping solution and choose **Solution Attributes**.

2. In the **Edit Solution** dialog box, in the **Data to Map** group, choose **Solid Temperatures** from the **Fields** list.

3. In the **Simulation Navigator**, right-click the **Constraint Set** and choose **New Constraint® Mapping Target Set**.

4. In the **Mapping Target Set** dialog box, select the **Destination Model Elements** or **Destination Model Nodes**.

Turbulent data flow results

What is it?

Turbulent data is a results option that generates post processing data evaluating turbulence at nodes for each fluid element.

> **Note**
>
> These results are only generated when either the **K-epsilon** option or the **Shear Stress Transport** option is selected from the **Viscous Model** list in the **Edit Solution** dialog box.

Slightly different results are generated, depending on the viscous model for the solution.

- The **K-epsilon** option generates Turbulence Energy data and Turbulence Dissipation data.

- The **Shear Stress Transport** option generates Turbulence Energy data and Specific Dissipation Rate data.

Turbulence Energy results characterize the amount of turbulence. The two dissipation results types (Turbulence Dissipation and Specific Dissipation Rate) characterize the lack of turbulence. Note that the two values indicate the opposite characteristics.

Why should I use it?

Use these results types to gain an understanding of the turbulence distribution in the flow field.

Where do I find it?

In Advanced Simulation, do the following to set up the solution to generate turbulent data results:

1. In the **Simulation Navigator**, right-click the active solution and choose **Edit Solution**.

2. In the **Edit Solution** dialog box, click the **Results Options** tab and select the **Turbulent data** check box.

Extract additional results

What is it?

A new button allows you to load additional results types for post-processing after the analysis is complete. Previously, you had to select all desired results types before running the analysis. To extract additional results after the analysis is complete, select the results type(s), and then click **Refresh** at the top of the Results Options page of the **Edit Solution** dialog box. This launches the results recovery module of the solver, and quickly recreates the results (.bun) file for post processing.

Why should I use it?

This feature makes it possible to first load only data of major interest into the results file. After post-processing these results, you may decide to discard your results and modify your model, or you may decide to extract additional results types from the same analysis. On the other hand, you may discover that you need additional results not foreseen when you selected results options on the **Edit Solution** dialog box before the analysis.

You avoid unnecessary processing time by loading only the results you know you need, yet you have quick access to all results without having to solving the model again.

Where do I find it?

In a simulation, with the solver type set to **NX Electronic Systems Cooling**, do the following to extract additional results after the analysis is complete:

1. In the **Post-Processing Navigator**, right-click the **Results** node and select **Close** to unload previous results.

2. In the **Simulation Navigator**, right-click the active solution and choose **Edit Solution**.

3. In the **Edit Solution** dialog box, on the Results Options page, select or clear any check box to add additional results, or remove any previously extracted results, and then click **Refresh** at the top of the page to regenerate the results.

4. In the **Simulation Navigator**, double-click the **Results** node to see the new results sets.

Flow velocity gamma report

What is it?

Velocity Gamma Values is a new option for the **Per Region** report type that provides scalar data equal to the velocity gamma at nodes for the selected elements. Velocity gamma is an index that measures flow uniformity. A Gamma value of unity indicates that the flow is totally uniform.

Why should I use it?

These results may be useful in determining whether or not a flow is fully developed, or to investigate flow uniformity in a specific part of the model.

Where do I find it?

In a solution, with the **Analysis Type** set to **Thermal-Flow**, do one of the following to create a **Per Region** type **Report**:

- In the **Simulation Navigator**, right-click **Simulation Objects** and choose **New Simulation Object® Report**. In the **Report** dialog box, select **Per Region**.

- On the **Advanced Simulation** toolbar, click **Simulation Object Type** **® Report** . In the **Report** dialog box, select **Per Region**.

Report results in spreadsheet format

What is it?

Some reported results specified in the **Report** simulation objects can now be generated as a comma separated value (.csv) file that can be opened in a spreadsheet program such as Microsoft Excel. The solver saves the file, groupReport.csv, in the run directory for the solution.

The results data stored in the file groupReport.csv are identical to the results data stored in the file groupReport.htm. See the NX 5.0 help for a full description of these results. Depending on the report type you create, the file contains the following data:

- **Between Regions —** Heat flow, view factors, radKs, and 3D flow pressure drop between selected regions of the model.

- **Per Region —** Temperature, heat load, heat flux, physical property, orbital and source view factors, phase change quality, and duct flow data.

- **Heat Map —** Heat load and heat flow into the selected elements via conduction, radiation, convection couplings, and thermal couplings.

Create two or more objects, and the Heat Map report generates heat flow data between each pair of **Report** objects.

- **Lift and Drag —** Lift and drag forces on selected surfaces. Typically, lift and drag vectors do not correspond to the model's global coordinate system, making it difficult to study these forces. You can generate a report detailing lift and drag forces at the center of gravity of each element in the selected region, isolating these vectors for flight applications.

Why should I use it?

Generating this data in tabular text format makes it easier to use other software such as a spreadsheet program for additional data processing not available in NX Advanced Simulation post-processing.

Where do I find it?

In a solution with solver type set to **NX Electronic Systems Cooling**, do the following two steps to generate the file groupReport.csv in the run directory:

1. Create a **Report** simulation object, selecting **Between Regions**, **Heat Map**, or **Lift and Drag** from the **Type** list.

2. Solve the model.

Solid properties check

What is it?

In an NX Electronic Systems Cooling analysis, you can use the new **Solid Properties Check** command to calculate the surface area for convection and radiation as well as the thermal capacitance of your model. See New model validation tools for more information.

NX Space Systems Thermal

Export primitives and radiation model

What is it?

In NX 5.0.1, you can export primitives and radiation models created with NX Space Systems Thermal to formats readable by other radiation analysis programs.

For EASRAD, TSS, and Thermica, you can export primitives preserving translation and rotation, relative element position and sequence, optical material properties, element thickness, and active (radiating) element face. Once imported into the other code, primitives originating in NX Space Systems Thermal behave the same as native primitives. See Export primitive overview for more information.

For TRASYS, SINDA, and ESATAN, you can export elements, physical properties and material properties. See Export radiation model overview for more information.

Where do I find it?

In Advanced Simulation, with a FEM file containing primitives open, do the following three steps to export primitives to one of the three supported radiation codes:

1. From the menu bar select **File** ® **Export** ® **Simulation**. The **Export** dialog box opens.

2. In the **Export** dialog box, select **Radiation Model** from the **File Type** list, and then select one of the four supported radiation codes from the **Radiation Model** list.

3. Depending on the radiation analysis program selected, type one or more file names in the **Name** field and click **OK**.

Report Per Region

What is it?

The data generated by the Report **Per Region** type appears in the HTML file GroupReport.htm in the run directory for the solution, and is saved to the spreadsheet file GroupReport.csv.

Select the Region of interest, then select one or more of the following:

- **Temperature** reports temperatures on the selected elements (average, maximum, minimum, and T_{max} - T_{min}), absorbed heat from different sources, and total heat absorbed.

- **Heat Load** reports total loads on selected elements, as well as heat loads broken down in terms of solar, diffuse solar, diffuse IR, and other types. The percentage of the total heat load on the model is also reported for the selected elements.

- **Heat Flux** reports total flux on selected elements, as well as heat flux broken down in terms of solar, diffuse solar, diffuse IR, and other types. The report is analogous to the Heat Loads report with values divided by area.

- **Physical Property** describes the selected elements in terms of total area, volume, mass and capacitance, as well as average density, thickness (shell elements), emissivity, absorptivity, conductivity, and specific heat.

- **Orbital and Source View Factors** reports Planet, Sun, and Albedo view factors for the selected elements.

- **Phase Change Quality** reports Average Quality, Maximum Quality, and Minimum Quality for the selected elements.

- **Duct Flow** reports Average Velocity, Average Pressure, Maximum Pressure, Minimum Pressure, Average Reynolds, and Average Mass Flow for selected elements.

Where do I find it?

In a solution with **Solution Type** set to **Space Systems Thermal**, do one of the following to create a **Report Per Region** type **Report**:

- In the **Simulation Navigator**, right-click **Simulation Objects** and choose **New Simulation Object® Report**. In the **Report** dialog box, select **Per Region**.

- First, on the **Advanced Simulation** toolbar, click **Simulation Object Type** **® Report** . In the **Report** dialog box, select **Per Region**.

Report Between Regions

What is it?

Report between regions allows the request of different solution data to be printed to the solution message file REPF. The **Report** can include thermal and flow data for elements, for regions of the model, or between regions of the model. Specialized report types allow you to track the flow solver results during the analysis, produce reports of lift and drag results, and create detailed heat maps.

The data generated by the **Between Regions** is generated by the solver and saved in the HTML file GroupReport.htm, and the in the run directory for the solution.

Select the **Primary Region** and the **Secondary Region**, and then select one or more of the options. The report contains data on the option(s) you select:

- **Heat Flow** processes and reports heat flowing from the **Primary Region** (i) to the **Secondary Region** (j). The temperature of the two sets of elements are reported, followed by heat flows broken down by Conduction, Radiation, Convection and Thermal Couplings, as well as the total heat flow.

- **View Factors and RadKs** post- processes and reports view factors from the **Primary Region** (i) to the **Secondary Region** (j). Emissivity values and area for both groups is reported, as well as black body and gray body view factors. Script F_{ij} is the gray body view factor, and $Radk_{ij}$ is s x ScriptF $_{ij}$.

Where do I find it?

In a solution with **Solution Type** set to **Space Systems Thermal**, do one of the following to create a **Between Regions** type **Report**:

- In the **Simulation Navigator**, right-click **Simulation Objects** and choose **New Simulation Object® Report**. In the **Report** dialog box, select **Between Regions**.
- On the **Advanced Simulation** toolbar, click **Simulation Object Type** **® Report** . In the **Report** dialog box, select **Between Regions**.

Solid properties check

What is it?

In an NX Space Systems Thermal analysis, you can use the new **Solid Properties Check** command to calculate the surface area for convection and radiation as well as the thermal capacitance of your model. See New model validation tools for more information.

Laminates post-processing

What is it?

Several enhancements to post-processing in the Composite Laminates tool extend its capabilities and make it easier to use.

- You can select all the solutions in the active simulation, or one or more solutions, subcases, or iterations as input for the post-processing. If you select a single solution, you can specify a single static subcase, mode or iteration.
- You have the option to use one or both of the following as input:
  - Solver shell stress resultants.
  - Solver ply stresses and strains.
- You can specify a safety factor for margin of safety computation.
- The post-processor generates one worksheet or csv file for each combination of solution, subcase or iteration, plus a global summary worksheet.
- The detailed results listing is now optional.
- You can optionally output solver shell stress resultant results, specifying which components to process.

- You can specify one, multiple or all the solutions in the active simulation.

Where do I find it?

To use the spreadsheet post-processor in the Laminates tool:

1. Load at least one simulation and FEM.

2. Load results in the **Post-Processing Navigator**. These can be one of your simulation solutions, or you can manually import any supported results file. There can be several loaded solutions, but at least one is required.

3. Click one of the results in the **Post-Processing Navigator** to display the Post View.

Motion Simulation

Load Transfer from Motion Simulation to Advanced Simulation

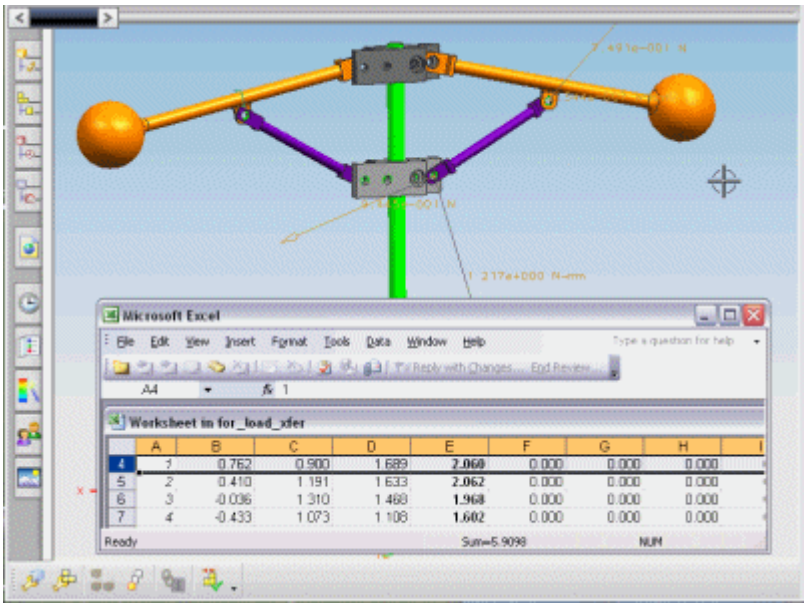
What is it?

You can now transfer mechanical loads at Motion Simulation joint locations for use as time-dependent loads in a finite element model in the Advanced Simulation application.

For a selected link, the **Load Transfer** command captures the reaction forces, torques, gravity, and link accelerations at each timestep in the simulation for each motion object (such as a joint) connected to the link. The captured forces and torques are saved directly in the Motion Simulation file and are also written to a spreadsheet for your reference. You can sort the spreadsheet to determine the timestep(s) where the highest reactions occur and then mark individual timesteps as *timesteps of interest*, for later reference in Advanced Simulation.

XY graphing records are also generated for each connected motion object at each timestep.

The animation below shows an example from Motion Simulation of the **Load Transfer** command capturing loads on two revolute joints in one of the part's support arms.



Load Transfer supports the following Motion Simulation objects:

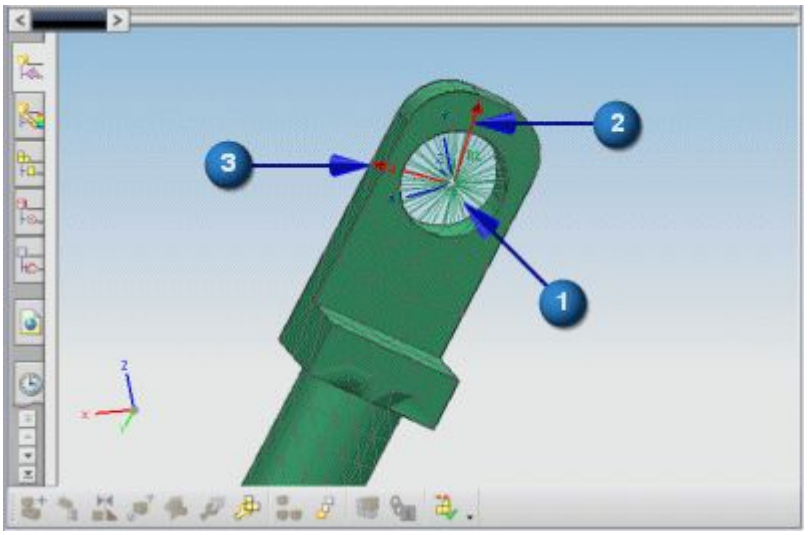
Supported	Not supported
All joint types	Curve-on-Curve
Spring	Point-on-Curve
Damper	Point-on-Surface
Bushing	Gear
Scalar force and torque	2D and 3D contact
Vector force and torque	Cable
	Rack and pinion

In Advanced Simulation, the reactions on a given motion object are imported in a time-dependent boundary condition Field as a force or moment. The load appears in the **Loads** container in the **Simulation Navigator**.

In the model, the load appears as a single node at the position of the motion object. The load's CSYS position and orientation correspond to the CSYS of the motion object that is relative to the link. The CSYS for a link acceleration corresponds to the inertia CSYS of the link. The gravity defined for the Motion Simulation solution is imported at the absolute CSYS.

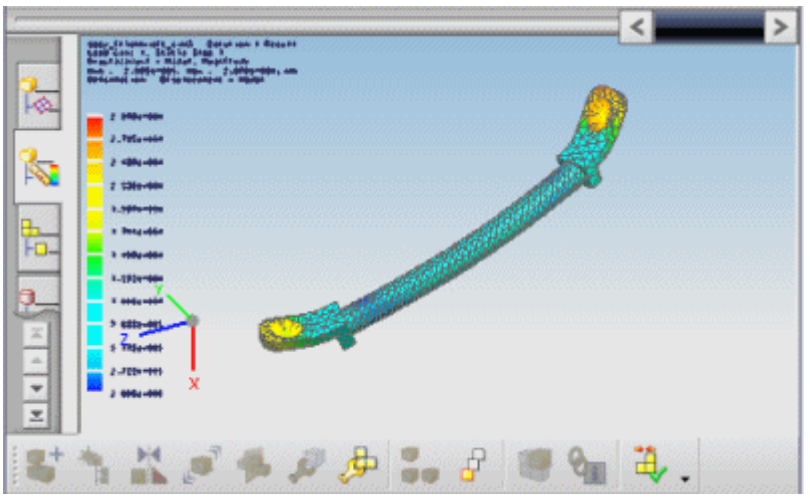
After meshing the part, you use a rigid spider element to connect the load transfer node to the mesh on the surrounding geometry.

The graphic below shows an example from Advanced Simulation of a spider element (1), an imported force (2), and moment (3) on the support arm.



In most cases, you will solve with the **Inertia Relief** solution attribute enabled. Before solving the model, you edit the Solution Attributes and specify a timestep of interest in the **Evaluation Time** box under **Boundary Condition Field Evaluation**.

The graphic below shows an example from Advanced Simulation of analyzing a load on the support arm at the location of the two joints.



Why should I use it?

Designers may have experience with Modeling and Motion Simulation, but not with the Advanced Simulation application. This enhancement allows designers to easily verify a product depending on the highest reaction values found during a motion simulation.

Where do I find it?

In Motion Simulation:

- On the **Motion** toolbar, click **Load Transfer** .

- Choose **Analysis® Motion® Load Transfer**.

In Advanced Simulation:

- In the **Simulation Navigator**, right-click the simulation node and choose **Import Motion Loads**.

- Choose **File® Import® Motion Loads**.

RecurDyn solver enhancements

What is it?

This release adds these enhancements to NX Motion Simulation and the RecurDyn solver.

- Export to Adams/View — You can now use the **Export to ADM** dialog box to export a mechanism from NX Motion Simulation that can be imported in MSC Adams/View. You can export the geometry in STL or Parasolid format.

- RecurDyn Variable Equation in motion driver functions — The RecurDyn solver now provides the Variable Equation (VE) to support state variables (such as displacements, forces, and so on) in functions for driving slider and revolute joints.

  This ability was previously available with Adams/Solver as the motion solver, but not with RecurDyn. NX Motion Simulation now fully supports the following variable statements with both solvers: ACCM, ACCX, ACCY, ACCZ, AX, AY, AZ, DM, DX, DY, DZ, FM, FX, FY, FZ, PHI, PITCH, PSI, ROLL, THETA, TM, TX, TY, TZ, VM, VX, VY, VZ, WDTM, WDTX, WDTY, WDTZ, WM, WX, WY, WZ, and YAW.

Where do I find it?

Export to Adams/View:

- Choose **File® Export® ADM**.

NX 5.0.2 Enhancements

Advanced Simulation

Supported solver versions

In this release, the software supports the following solver versions:

- NX Nastran 5 and earlier versions
- MSC Nastran 2007 and earlier versions
- ANSYS 11 and earlier versions
- ABAQUS 6.7-1 and earlier versions

NX Thermal and Flow

Batch solving

What is it?

Batch solving allows you to analyze any number of solver input files with a single solve command. You can set up batch solving by creating a specific text file and saving it in a specified location.

Why should I use it?

This functionality eliminates unnecessary user input in what is essentially a series of software operations. It allows multiple analysis jobs to be run in sequence without human intervention.

Where do I find it?

To set up batch analysis runs, do the following:

1. Create a solver input file for each analysis, noting its file name and location.
2. Using a text editor, create an ASCII text file that lists the input files in the order in which they should be solved. Include the full path with each input file name, and use a new line for each input file path/name.
3. Name the text file *multipleruns.dat* and save it in the same directory as the first input file in the list.
4. Solve the input file for the first analysis. The other input files listed in the text file are automatically solved in sequence.

Support for inherited materials

What is it?

You can use inherited materials to define a mesh collector. The solver correctly interprets the intended material and includes this data in the analysis. Previously you had to explicitly define the material for the mesh collector.

Why should I use it?

This allows you to propagate material properties on all meshes associated with given geometry.

Where do I find it?

To define a Mesh Collector with inherited material properties: on the **Mesh Collector** dialog box, select **Inherited** from the **Material** list.

Internal thermal boundary conditions

What is it?

You can define a thermal boundary condition directly on a polygon face embedded in the fluid without having to create a 2D mesh or an embedded flow surface. The heat load is applied on the fluid faces. No flow surface is created and the selected surface will therefore not block the fluid. The thermal boundary condition automatically transfers heat to the fluid using default convection properties defined on the Ambient Conditions page of the **Edit Solution** dialog box.

Why should I use it?

This functionality makes it possible to apply thermal boundary conditions to model geometry embedded in the flow domain without blocking the flow. A typical application is a coarse screen or other device that convects heat to the fluid but does not impede flow.

Periodic pressure differential

What is it?

When defining a periodic boundary condition, you can specify a pressure rise or drop between two sets of periodic faces. In this case, velocity is periodic, but temperature and pressure are not. Create this type of periodic boundary condition in the usual way, but in the **Name** box of the periodic boundary condition dialog box, type `_#PRESSURE#XXX.XX#`. In place of `XXX.XX`, type the actual pressure value, using the current model units. The solver uses the initial 3D flow pressure as a reference for scale. The pressure reference value

defaults to the value specified in ambient conditions. Alternately, it can be specified by defining a pressure value for 3D Flow Initial Conditions.

Pressure results accurately show the distribution of relative pressure differential throughout the flow domain. The physical validity of pressure results depends on the accuracy of the pressure you specify as an initial condition. If the initial value you specify accurately models physical conditions in a given periodic volume, the analysis produces physically valid pressure results for that volume.

Why should I use it?

Use this feature when the flow over a large array of objects must be simulated by isolating the air volume around a single instance of the object. An example application is the simulation of television tubes being cured in a long oven as part of the manufacturing process.

Where do I find it?

To open the **Periodic Boundary Condition** dialog box, do one of the following in an **Advanced Flow** solution or an **Advanced Thermal-Flow** solution:

- In the **Simulation Navigator**, right-click **Simulation Objects** and choose **New Simulation Object**® **Periodic Boundary Condition**.
- On the **Advanced Simulation** toolbar, click **Simulation Object Type** , and then click **Periodic Boundary Condition** .

Import CGNS and PLOT3D fluid mesh

What is it?

You can import a fluid mesh in CGNS format or PLOT3D format into an NX Flow FE model. Only elements are imported. Boundary conditions and element properties are not imported.

Where do I find it?

To import a CGNS or PLOT3D fluid mesh, do the following.

1. Choose **File**® **Import**® **Simulation**.
2. In the **Import** dialog box, select either **NX THERMAL/FLOW** or **NX ELECTRONIC SYSTEMS COOLING**, and then click **OK**.
3. In the **Import Simulation** dialog box, select either **CGNS** or **PLOT3D** from the **File Type** list.
4. Click **Browse** to identify the **Input File** for import.

5. Click **OK** to import the mesh.

Automatic limiter for advection calculations

What is it?

For second order advection schemes, the solver now uses an automatic limiter function for advection calculations.

By default, the following advection calculation types are controlled by the automatic limiter:

- **Momentum**
- **Energy**
- **Two-Equation Turbulent Model**
- **Scalars and Humidity**

At each iteration, the solver calculates and applies the optimal limiter value for each advection calculation type at each control volume face. This limits any non-physical overshoots which may be produced by the second order scheme.

You can still use the previous method, which is to specify a fixed limiter value for each advection calculation type, or accept the default. To use the previous method:

1. Right-click the solution and choose **Solver Parameters**.
2. On the **Solver Parameters** dialog box, select the 3D Flow Solver tab, then expand the **Advection Schemes** group.
3. From one of the four lists (**Momentum**, **Energy**, **K-Epsilon**, or **Scalars and Humidity**) choose either **Second order (QUICK)** or **Second order (SOU)**.
4. From the associated **Limiter** list, choose **Specify**, and enter a limiter value in the box that appears.

Why should I use it?

The automatic limiter function results in more accurate results without unnecessarily increasing solution time.

Where do I find it?

The automatic limiter is active by default whenever you use a second-order advection scheme.

Absolute or relative pressure mapping option

What is it?

When defining what type of pressure values to map to a target model, you can select either absolute pressure values or pressure values that are relative to ambient. Previously you could only map pressure values that were relative to ambient.

You specify the type of pressure values to map as a solution attribute.

Why should I use it?

Absolute pressures are required if local deformations of a cavity-type structure are calculated. If relative pressures are used, ambient pressure is assumed inside the cavity.

Where do I find it?

To specify that values mapped to the target model are absolute pressure values, do the following in a **Mapping** analysis type:

1. Right-click the solution and choose **Edit Attributes**.

2. On the Mapping details page of the **Edit Solution** dialog box, in the **Data to Map** group, select **Flow Values** from the **Fields** list.

3. Select **Absolute** from the **Pressure** list.

NX Electronic Systems Cooling

ESC Advanced Flow Features

What is it?

The solution type **NX Advanced Thermal/Flow with ESC** adds advanced flow functionality to NX Electronic Systems Cooling. The new solution type replaces the **Electronic Systems Cooling — Advanced** solution type, and includes all the capabilities formerly provided by this type. Several new flow features are added.

Moving Frame of Reference simulation object

A moving frame of reference is a CFD technique for modeling the impact of rotating or translating machinery or watercraft on the surrounding inertial fluid. You can model two types of motion. Use a **Rotating Frame of Reference** to model the fluid perturbations of rotating machinery such as impellers or turbine blades spinning at a constant rate in an inertial fluid. Use a **Translating Frame of Reference** to model fluid perturbations caused by the movement of an object such as a hull or hydrofoil as it passes through an inertial fluid at a constant velocity.

Periodic Boundary Condition simulation object

A periodic boundary condition forces the flow and scalar fields to be identical at the periodic boundaries, resulting in a spatially cyclic solution. The sections can be identical translationally or rotationally. The periodicity is both fluid and thermal. If the application can be modeled as a series of identical joined sections, you need construct and solve only one section, potentially greatly reducing model preparation and solution times in certain applications.

Humidity option

The flow solver models humidity with a general scalar equation. It traces the movement of water vapor through the fluid domain, updating the density, specific heat at constant pressure (Cp), thermal conductivity, and dynamic viscosity. You can define humidity by specifying relative or specific humidity values for air entering the flow domain, or as an initial condition for the flow analysis.

Scalar modeling object

You can set up the simulation of mixing and diffusing of one or more fluids or fluid-like components with the main fluid of the 3D flow simulation. You can model two types of scalar mixtures, **Passive** and **General Gas**. You can define scalar mixtures by specifying values for the fluid entering the flow domain, or as an initial condition for the flow domain, or for regions of the flow domain.

Vorticity, **Humidity**, and **Scalar** results types

These are available as options for the **Selective Results** simulation object and as Results Options in the **Edit Solution** dialog box. Vorticity results measure turbulence intensity. Humidity results measure the relative humidity and specific humidity. Scalar results measure scalar to mixture mass fractions.

High Speed Flow solution attribute

The **High Speed Flow** option on the **Edit Solution** dialog box activates the total energy equation to handle compressible flow problems. The solver accurately models subsonic, sonic and transonic conditions. Best accuracy is achieved at speeds below Mach 4.

Supersonic Inlet

This simulation object allows modeling of fluid entering the domain at supersonic velocity. As with an **Inlet Flow** type flow boundary condition, you define a Supersonic Inlet on a bounding surface of the fluid domain, adjacent to the fluid elements. You define flow in terms of Mach number (velocity / speed of sound)

Non-Newtonian Fluid modeling object

You can include a non-Newtonian fluid in the model by modifying the definition of a fluid material to include properties required for such a fluid in the Power-Law model or the Herschel-Bulkley model. The modified material, designated the **Affected Fluid Material**, and the non-Newtonian fluid properties, must be specified in a **Non-Newtonian Fluid** modeling object. Wherever the affected fluid material appears in the model, the solver interprets it as having the non-Newtonian properties you define. Non-Newtonian fluid modeling has diverse applications such as simulating molten polystyrene, blood, nuclear fuel slurry, and toothpaste.

Where do I find it?

The advanced flow features are only available in the solution type **NX Advanced Thermal/Flow with ESC**. To create this solution type:

1. In a simulation, do one of the following:
   - In the **Simulation Navigator**, right-click the simulation and choose **New Solution**.
   - Choose **Insert® Solution**.
   - On the Advanced Simulation toolbar, select **Solution** .
2. In the **Create Solution** dialog box, do the following:
   - Select **NX ELECTRONIC SYSTEMS COOLING** from the **Solver** list.
   - Select **Coupled Thermal-Flow** from the **Analysis Type** list.
   - Select **NX Advanced Thermal/Flow with ESC** from the **Solution Type** list.

Batch solving

What is it?

Batch solving allows you to analyze any number of solver input files with a single solve command. You can set up batch solving by creating a specific text file and saving it in a specified location.

Why should I use it?

This functionality eliminates unnecessary user input in what is essentially a series of software operations. It allows multiple analysis jobs to be run in sequence without human intervention.

Where do I find it?

To set up batch analysis runs, do the following in a solution:

1. Create a solver input file for each analysis, noting its file name and location.

2. Using a text editor, create an ASCII text file that lists the input files in the order in which they should be solved. Include the full path with each input file name, and use a new line for each input file path/name.

3. Name the text file *multipleruns.dat* and save it in the same directory as the first input file in the list.

4. Solve the input file for the first analysis. The other input files listed in the text file are automatically solved in sequence.

Support for inherited materials

What is it?

You can use inherited materials to define a mesh collector. The solver correctly interprets the intended material and includes this data in the analysis. Previously you had to explicitly define the material for the mesh collector.

Why should I use it?

This allows you to propagate material properties on all meshes associated with given geometry.

Where do I find it?

To define a Mesh Collector with inherited material properties: on the **Mesh Collector** dialog box, select **Inherited** from the **Material** list.

Internal thermal boundary conditions

What is it?

You can define a thermal boundary condition directly on a polygon face embedded in the fluid without having to create a 2D mesh or an embedded flow surface. The heat load is applied on the fluid faces. No flow surface is created and the selected surface will therefore not block the fluid. The thermal boundary condition automatically transfers heat to the fluid using default convection properties defined on the Ambient Conditions page of the **Edit Solution** dialog box.

Why should I use it?

This functionality makes it possible to apply thermal boundary conditions to model geometry embedded in the flow domain without blocking the flow. A typical application is a coarse screen or other device that convects heat to the fluid but does not impede flow.

Periodic pressure differential

What is it?

When defining a periodic boundary condition, you can specify a pressure rise or drop between two sets of periodic faces. In this case, velocity is periodic, but temperature and pressure are not. Create this type of periodic boundary condition in the usual way, but in the name box of the periodic boundary condition dialog box, type `_#PRESSURE#XXX.XX#`. In place of `XXX.XX`, type the actual pressure value, using the current model units. The solver uses the initial 3D flow pressure as a reference for scale. The pressure reference value defaults to the value specified in ambient conditions. Alternately, it can be specified by defining a pressure value for 3D Flow Initial Conditions.

Pressure results accurately show the distribution of relative pressure differential throughout the flow domain. The actual values of the pressure results accurately model physical conditions in a given periodic volume in so far as the pressure you define as an initial condition accurately models the pressure in the domain for that periodic volume.

Why should I use it?

You should use this feature when the flow over a large array of objects must be simulated by isolating the air volume around a single instance of the object. An example application is the simulation of television tubes being cured in a long oven as part of a manufacturing process.

Where do I find it?

To open the **Periodic Boundary Condition** dialog box, do one of the following in an **Advanced Thermal / Flow with ESC** solution:

- In the **Simulation Navigator**, right-click **Simulation Objects** and choose **New Simulation Object® Periodic Boundary Condition**.
- On the **Advanced Simulation** toolbar, click **Simulation Object Type** , and then click **Periodic Boundary Condition** .

Import CGNS and PLOT3D fluid mesh

What is it?

You can import a fluid mesh in CGNS format or PLOT3D format into an NX Electronic Systems Cooling FE model. Only elements are imported. Boundary conditions and element properties are not imported.

Where do I find it?

To import a CGNS or PLOT3D fluid mesh, do the following.

1. Choose **File® Import® Simulation**.
2. In the **Import** dialog box, select either **NX THERMAL/FLOW** or **NX ELECTRONIC SYSTEMS COOLING**, and then click **OK**.
3. In the **Import Simulation** dialog box, select either **CGNS** or **PLOT3D** from the **File Type** list.
4. Click **Browse** to identify the **Input File** for import.
5. Click **OK** to import the mesh.

Automatic limiter for advection calculations

What is it?

For second order advection schemes, the solver now uses an automatic limiter function for advection calculations.

By default, the following advection calculation types are controlled by the automatic limiter:

- **Momentum**
- **Energy**
- **Two-Equation Turbulent Model**

- **Scalars and Humidity**

At each iteration, the solver calculates and applies the optimal limiter value for each advection calculation type at each control volume face. This limits any non-physical overshoots which may be produced by the second order scheme.

You can still use the previous method, which is to specify a fixed limiter value for each advection calculation type, or accept the default. To use the previous method:

1. Right-click the solution and choose **Solver Parameters**.

2. On the **Solver Parameters** dialog box, select the 3D Flow Solver tab, then expand the **Advection Schemes** group.

3. From one of the four lists (**Momentum**, **Energy**, **K-Epsilon**, or **Scalars and Humidity**) choose either **Second order (QUICK)** or **Second order (SOU)**.

4. From the associated **Limiter** list, choose **Specify**, and enter a limiter value in the box that appears.

Why should I use it?

The automatic limiter function results in more accurate results without unnecessarily increasing solution time.

Where do I find it?

The automatic limiter is active by default whenever you use a second-order advection scheme.

Absolute or relative pressure mapping option

What is it?

When defining what type of pressure values to map to a target model, you can select either absolute pressure values or pressure values that are relative to ambient. Previously you could only map pressure values that were relative to ambient.

You specify the type of pressure values to map as a solution attribute.

Why should I use it?

Absolute pressures are required if local deformations of a cavity-type structure are calculated. If relative pressures are used, ambient pressure is assumed inside the cavity.

Where do I find it?

To specify that values mapped to the target model are absolute pressure values, do the following in a **Mapping** analysis type:

1. Right click the solution and choose **Edit Attributes**.

2. On the Mapping details page of the **Edit Solution** dialog box, in the **Data to Map** group, select **Flow Values** from the **Fields** list.

3. Select **Absolute** from the **Pressure** list.

NX Space Systems Thermal

Batch solving

What is it?

Batch solving allows you to analyze any number of solver input files with a single solve command. You can set up batch solving by creating a specific text file and saving it in a specified location.

Why should I use it?

This functionality eliminates unnecessary user input in what is essentially a series of software operations. It allows multiple analysis jobs to be run in sequence without human intervention.

Where do I find it?

To set up batch analysis runs, do the following:

1. Create a solver input file for each analysis, noting its file name and location.

2. Using a text editor, create an ASCII text file that lists the input files in the order in which they should be solved. Include the full path with each input file name, and use a new line for each input file path/name.

3. Name the text file *multipleruns.dat* and save it in the same directory as the first input file in the list.

4. Solve the input file for the first analysis. The other input files listed in the text file are automatically solved in sequence.

Support for inherited materials

What is it?

You can use inherited materials to define a mesh collector. The solver correctly interprets the intended material and includes this data in the analysis. Previously you had to explicitly define the material for the mesh collector.

Why should I use it?

This allows you to propagate material properties on all meshes associated with given geometry.

Where do I find it?

To define a Mesh Collector with inherited material properties: on the **Mesh Collector** dialog box, select **Inherited** from the **Material** list.

Export elements as primitives

What is it?

When you export a radiation model from NX Space Systems Thermal, NX exports individual elements in the FEM as primitives. Previously, primitives were exported but not elements. NX exports each element in the FE model as a separate primitive.

Where do I find it?

To export a radiation model file, do the following in an NX Space Systems Thermal FE model or simulation:

1. Choose **File**® **Export**® **Simulation**.

2. In the **Export** dialog box, select **Radiation Model** from the **File Type** list.

Motion Simulation

New RecurDyn 3D contact method

What is it?

A new 3D contact method called **Fitted** has been introduced that significantly improves RecurDyn Solver 3D contact performance. The **Fitted** contact method creates a local smooth surface that is fitted to facets in order to improve contact smoothness and simulation speed. This method takes advantage of the following:

- Optimized transfer of surface data from NX to the RecurDyn solver

- A fast collision-detection algorithm
- "Noise" reduction for greater stability when surfaces are in continuous contact, which results in no oscillation of the penetration and a smoother reaction

For NX 5.0.2, there are now two RecurDyn 3D contact methods: **Fitted** and **Faceted**. **Faceted** is the preexisting NX 5 RecurDyn contact method and is similar to the Faceted 3D contact method used by the Adams/Solver.

Why should I use it?

The new **Fitted** contact method improves the simulation performance of 3D contact, while also improving the smoothness of the impact and sliding contact.

- For best results between convex surfaces (surfaces that are curved or rounded outward), use the **Fitted** contact method.
- For best results between flat or concave surfaces (surfaces that are curved inward, like a segment of the interior of a circle or hollow sphere), use the **Faceted** contact method.

Where do I find it?

In Motion Simulation:

1. Choose **File**® **Utilities**® **Customer Defaults**.
2. Select **Motion**® **Extras**.

Faceted and **Fitted** are on the RecurDyn 3D Contact page, under **Contact Method**.

NX 5.0.3 Enhancements

NX Thermal and Flow

Fluid Domain mesh quality check

Fluid Domain mesh size optimization

What is it?

The new **Allow mesh size variations inside fluid domains** option in the **Create Solution** and **Edit Solution** dialog boxes lets you optimize the size of elements in a fluid mesh generated with the **Fluid Domain** simulation object command. If you select this new option, the software increases the element size towards the center of the volume.

Why should I use it?

Larger elements in open areas of the fluid mesh reduce model size and solution time but still provide good accuracy.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Thermal and Flow	Flow	Flow or Advanced Flow
	Coupled Thermal-Flow	Thermal-Flow, Advanced Thermal-Flow, or Complete

Where do I find it?

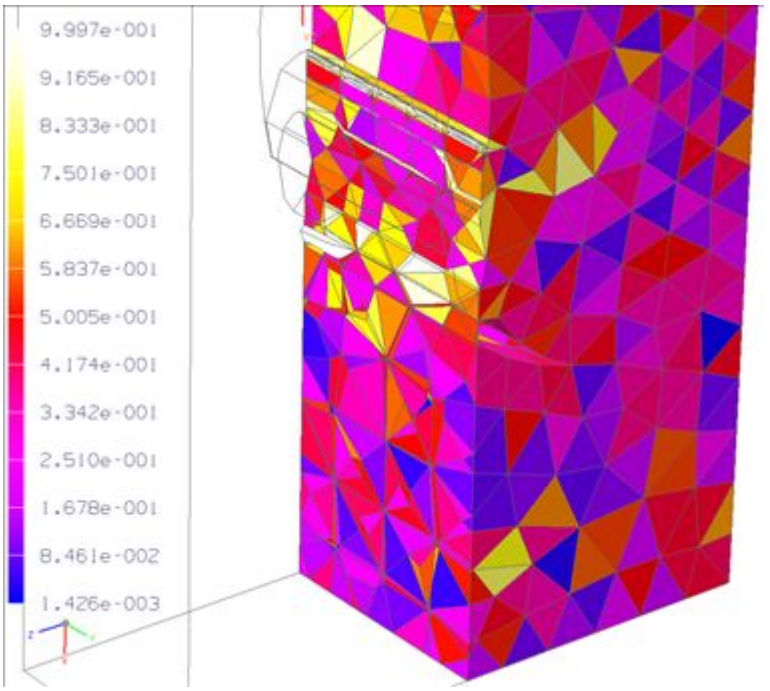
Application	Advanced Simulation
Prerequisite	A Simulation file with an active solution that has the appropriate solver, analysis type, and solution type selected.
Resource bar	In the Simulation Navigator, right-click the solution name and choose® **Solution Attributes**.
Dialog box	3D Flow page
Location on page	**Fluid Domain Parameters**® **Allow mesh size variations inside fluid domains**.

Fluid Domain mesh quality check

What is it?

You now can check the element quality of a fluid mesh generated with the **Fluid Domain** simulation object command. When you build an input file for a solution that uses a **Fluid Domain** simulation object to create a fluid mesh, the software generates three element quality results sets: **Element Size**, **Element Aspect Ratio**, and **Element Skew**. You can display these results sets in the Post-Processing Navigator to evaluate the quality of the elements in the fluid mesh.

- Use **Element Size** results to locate elements that are too small or too large.
- Use **Element Aspect Ratio** and **Element Skew** results to locate malformed elements that may cause convergence and accuracy problems.



Where do I find it?

Application	Advanced Simulation
Prerequisite	Input file generated from an NX Thermal and Flow solution that contains a **Fluid Domain** simulation object.
Resource bar	**Post-Processing Navigator**® [simulation name]® [solution name]
Results set	**Element Size**, **Element Aspect Ratio** or **Element Skewness**

NX Electronic Systems Cooling

Fluid Domain mesh quality check

Fluid Domain mesh size optimization

What is it?

The new **Allow mesh size variations inside fluid domains** option in the **Create Solution** and **Edit Solution** dialog boxes lets you optimize the size of elements in a fluid mesh generated with the **Fluid Domain** simulation object command. If you select this new option, the software increases the element size towards the center of the volume.

Why should I use it?

Larger elements in open areas of the fluid mesh reduce model size and solution time but still provide good accuracy.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Electronic Systems Cooling, NX Advanced Thermal/Flow with ESC

Where do I find it?

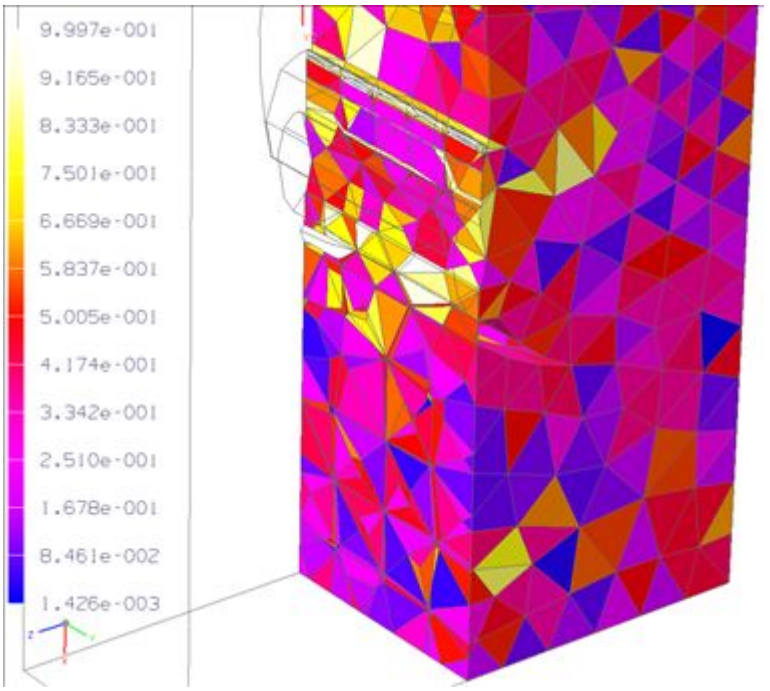
Application	Advanced Simulation
Prerequisite	A Simulation file with an active solution that has the appropriate solver, analysis type, and solution type selected.
Resource bar	In the Simulation Navigator, right-click the solution name and choose® **Solution Attributes**.
Location in dialog box	3D Flow page
Location on page	**Fluid Domain Parameters® Allow mesh size variations inside fluid domains**.

Fluid Domain mesh quality check

What is it?

You now can check the element quality of a fluid mesh generated with the **Fluid Domain** simulation object command. When you build an input file for a solution that uses a **Fluid Domain** simulation object to create a fluid mesh, the software generates three element quality results sets: **Element Size**, **Element Aspect Ratio**, and **Element Skew**. You can display these results sets in the Post-Processing Navigator to evaluate the quality of the elements in the fluid mesh.

- Use **Element Size** results to locate elements that are too small or too large.
- Use **Element Aspect Ratio** and **Element Skew** results to locate malformed elements that may cause convergence and accuracy problems.



Where do I find it?

Application	Advanced Simulation
Prerequisite	Input file generated from an NX Electronic Systems Cooling solution that contains a **Fluid Domain** simulation object.
Resource bar	**Post-Processing Navigator**® [simulation name]® [solution name]
Results set	**Element Size**, **Element Aspect Ratio**, or **Element Skew**

NX Space Systems Thermal

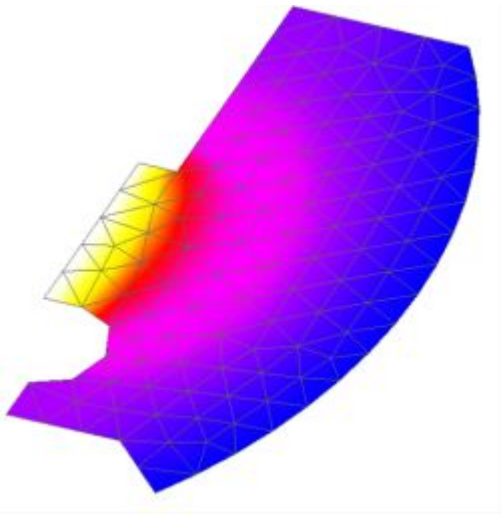
Axisymmetric modeling

What is it?

You can now use axisymmetric modeling techniques. Two new solution types, **Axisymmetric Thermal**, and **Advanced Axisymmetric Thermal**, provide a range of boundary conditions and solution attributes applicable to axisymmetric modeling. If you have a license for NX Space Systems Thermal, you can work with these new solution types by creating a new solution where you specify **NX THERMAL/FLOW** as the **Solver** and **Axisymmetric Thermal** as the **Analysis Type** in the **Create Solution** dialog box.

Axisymmetric modeling reduces a 3D model that is symmetrical about an axis to a 2D model that generates equivalent results more quickly. You can create an axisymmetric model only if the physical model is:

- Geometrically axisymmetric.
- Materially axisymmetric.
- Thermally axisymmetric.



Why should I use it?

For appropriate models, axisymmetric modeling:

- Simplifies model preparation.
- Can greatly reduce analysis time.
- Does not sacrifice detail or accuracy.

Where do I find it?

Application	Advanced Simulation
Prerequisite	A Simulation file, with an active solution that has **NX Thermal/Flow** as the solver, **Axisymmetric Thermal** as the analysis type, and either **Axisymmetric Thermal** or **Advanced Axisymmetric Thermal** as the solution type.
Resource bar	**Simulation Navigator**® [right click simulation]® **New Solution**

Motion Simulation

Export RecurDyn input file

What is it?

You can now export your Motion Simulation model and geometry to a set of input files that can be imported directly into RecurDyn Professional.

The export process creates three files:

.xmt_txt	Parasolid file that contains the CAD geometry.
.rbx	RecurDyn binary XML file that includes the Motion Simulation model information.
.sdk	Geometry relation file that associates the geometry objects with the motion objects (such as links and contacts). This file references the contents of the .xmt_txt and .rbx files.

After exporting the input files from NX, you can import the .sdk file into RecurDyn Professional, where you can edit the model and solve it as needed.

Where do I find it?

Application	Motion Simulation
Prerequisite	Simulation file active
Menu	**File® Export® RecurDyn Input**
Shortcut menu	Right-click the simulation file® **Export® RecurDyn Input**
Location in dialog box	In the **Files of Type** list, choose **RecurDyn SDK File (*.sdk)**

Product Validation

Check-Mate

DFM Advisor Checker

What is it?

Check-Mate has 15 additional DFM checking criteria. You can now:

- Validate part design for manufacturing during the modeling process.
- Run multiple parts in a single automated process.
- Track checks that enforce TcAE Validation Master Form enforcement and workflow.

- Automate, communize, and enforce the design for the manufacturing process.

The DFM checking criteria are added under three categories:

Trim Category:

- **Typical Cutouts** — Checks if the cutout is within the specified up and down angle limits and between the minimum and maximum sizes. Cutouts that are larger or smaller than specified limits require additional die maintenance and construction costs.

- **Trim to Trim** — Checks the distance from cutout to cutout, to analyze whether additional operations or additions of a mounting foot, higher quality pad, and trim steel materials are required to accomplish the necessary trimming.

- **Trim to Trim Perimeter** — Checks the distance from cutout to perimeter or from perimeter to perimeter to analyze whether additional operations or additions of a mounting foot, higher quality pad, and trim steel materials are required to accomplish the necessary trimming.

Pierce Category:

- **Typical Hole** — Checks if the hole has a circumscribed shape less than the Pierce Limit, and if the hole is manufactured using a single punch, to ensure that standard pierce equipment can be used.

- **Flanged Hole** — Checks if a flanged hole has a continuous flange around a circle or an oblong, to ensure that the flange hole shape can be manufactured using standard pierce equipment.

- **Hole Size** — Checks if a standard size is used for a standard shape (for example, round, square), to reduce the number of unique pierce punches used in a die.

- **Hole Corner Radii** — Checks whether pierce shapes have the correct corner radii size to meet manufacturing conditions.

- **Pierce Commonization** — Compares all pierce types (gage, critical, or standard), hole sizes, shank sizes, and shapes for each die, and reports objects that are not common, to identify identical hole sizes, pierce types, or shapes for each punch shank diameter used within a single die.

- **Hole to Trim** — Checks the distance of a hole (pierce object) from a trim object, to determine if a pierce object can be manufactured using standard pierce equipment.

- **Hole to Hole** — Checks the distance between the neighboring holes (pierce object). Checks the distance between neighboring holes (pierce objects), to determine if they can be manufactured using standard pierce equipment. The piercing of a hole requires a minimum amount of clearance around the hole.

Flange Category:

- **Flange Break Radius** — Checks the flange radius (the radius the metal is bent around), to determine if it is too big or too small. Either of these conditions adds die development costs and/or die maintenance costs.

- **Flange Flat Length** — Checks the distance from the end of the break radius to the end of the flange, to ensure that the flat length is large enough for the use of standard steel flanges and the part can be positioned correctly in the die.

- **Flange to Trim** — Checks the distance from the flange tangency curve to the nearest trim object to determine if the trim line is too close to a flange.

- **Flange to Hole** — Checks if distance from the flange tangency curve to the hole is greater than the minimum value for this criteria, to determine if flanging can be accomplished with standard equipment or if hole distortion is expected from flanging too close to a pierce shape.

- **Flange to Flange** — Checks the distance between two opposing flanges, to determine if they are too close to be manufactured. This check guards against flanges being too close, resulting in potentially weak flanging steels.

Why should I use it?

Use the **DFM Advisor Checkers** to check the manufacturability of your design to determine whether it meets corporate manufacturing standards.

Where do I find it?

To run the checks:

1. Choose **Analysis® Check-Mate® Run Tests**.

   Or, on the **Check-Mate** toolbar, click **Run Tests** .

2. Then, in the **Rule Tests** dialog box, select the DFM checks from **DFM Category** and add them to the selected tests list, and execute the check.

Check Flag enhancement

What is it?

Check-Mate now includes the following Smart Checking options on the Run Options page:

- **Generate Check Flag** — Creates a part attribute representing the **Check Flag** when the checking results are saved to the part file. The part attribute **Check Flag** is not created if the part file is not saved with the checking results.

- **Skip Checking if Check Flag is up-to-date** — Check-Mate reads the **Check Flag**, and if the flag is valid, the checker is not run. If the **Check Flag** does not exist in the part file, the part file is tested with the checker. For a profile containing many child checkers, there is only one check flag for the profile, and there are none for the child checkers. If the profile is to be run, all of its child checkers must run.

- **Read Flag without part loading** — Check-Mate reads the **Check Flag** from the part file without loading it, and if Check-Mate finds the flag existing in the part, it assumes the flag is up-to-date and the checker is not run. If the Check Flag does not exist in the part file, the part file is tested with the checker.

  Note

  The **ug_check_part** command line utility also includes these Smart Checking options.

You can also set the following environment variables:

- **UGCHECKMATE_GENERATE_CHECK_FLAG** [True | False] — The default is False which means do not create a Part Attribute for Check Flag.

- **UGCHECKMATE_SKIP_CHECK_IF_FLAG_UP_TO_DATE** [True | False] — The default is False which means all the specified checkers are run.

- **UGCHECKMATE_READ_FLAG_WITHOUT_PART_LOADING** [True | False] — The default is False which means the part is loaded in order to determine the valid status of the Check Flag.

Why should I use it?

You can greatly improve checking performance by generating a check flag and skipping checking when the flag is valid.

Where do I find it?

Choose **Analysis→Check-Mate→Run Tests**.

Choose **File→Utilities→Customer Defaults**. From the Analysis list, select **Extras**. Click the Check-Mate Run Options tab.

Show Flags dialog box enhancements

What is it?

The Check-Mate **Show Flags** dialog box now shows the result level and status of Check Flags in the part. It also detects whether the part attribute which is associated with **Check Flag** has been modified outside of Check-Mate.

Why should I use it?

The new enhancement allows you to view the result level and status of the **Check Flag**.

Where do I find it?

Choose **Analysis→Check-Mate->Show Flags**.

Manufacturing Design Tools

Weld Assistant

Fillet Weld

What is it?

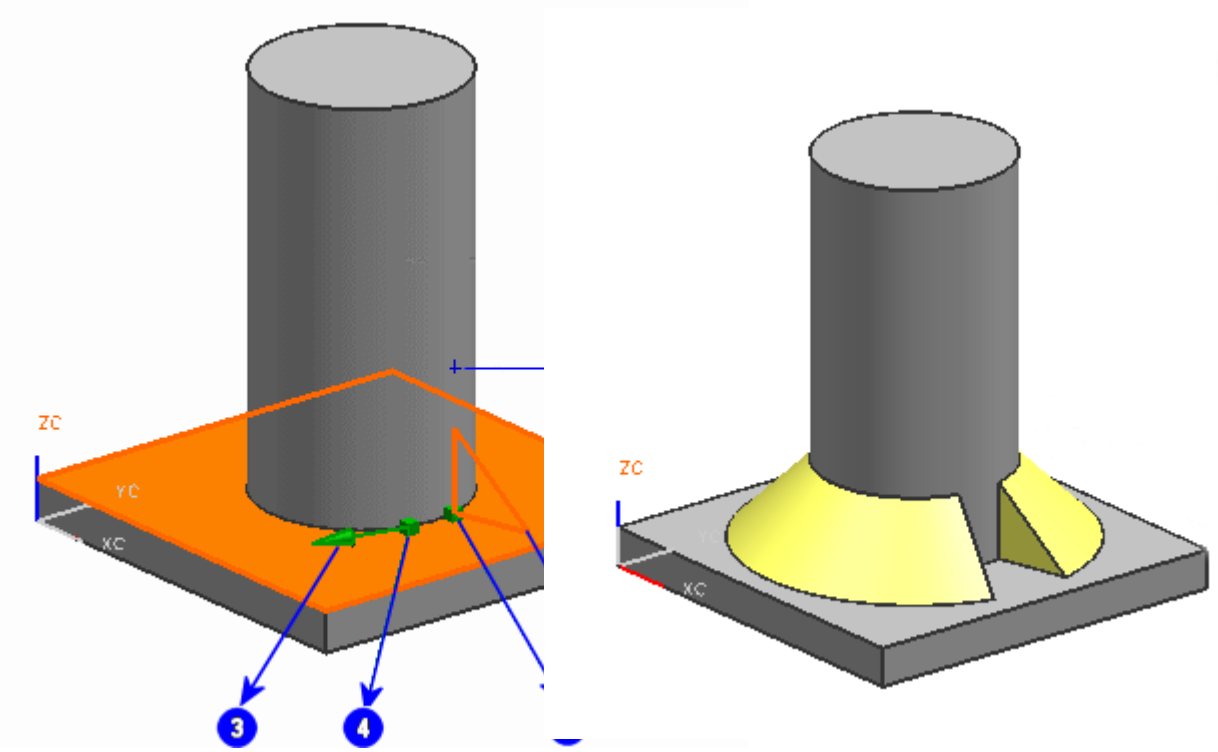
The **Fillet** command is part of Weld Assistant that enables the authoring of welds. Use this command to define the fillet arc weld in 3D model space to join multiple components together.

The **Fillet** command has the following enhancements:

- Selection Intent is available to aid face selection.
- When you select both the face sets to create the fillet weld, start and end limit handles are displayed in the graphics window. You can drag these handles to change the start or end positions, or you can directly type the required value in the input box in the graphics window.
- A preview of the fillet weld cross section is available. The location of the preview image is controlled by the location of the first selected face. Points are displayed to indicate the selected location.

Before creation

After creation



1. Point where the first face is selected
2. Point where the second face is selected
3. Direction of the fillet weld
4. Start point of the fillet weld
5. End point of the fillet weld
6. Preview of the fillet weld cross section

Fillet weld example

- You can manually type the required extension value when you need to extend faces that do not intersect the opposite faces.
- You can create the fillet weld from the start point to the end point, or from the end point to the start point using the **Create End to Start** option. This option controls creating a portion of a fillet weld when the selected faces are closed. This is important if the fillet weld crosses the boundary start point.
- You can recreate an individual skip weld you deleted using the **Recreate Deleted Skip Welds** option. This option is available only when you edit a skip weld.

Where do I find it?

In the Modeling application, do one of the following:

- Choose **Insert® Welding® Fillet**.
- On the **Weld Assistant** toolbar, click **Fillet** .

Weld Point

What is it?

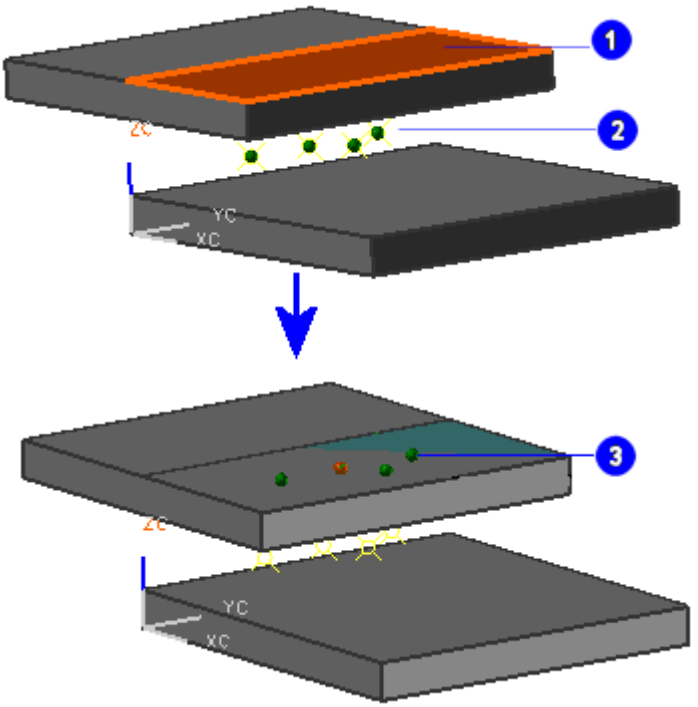
The **Weld Point** enhancements enable you to create weld points by specifying 2D values and then projecting them along a vector to a reference sheet. You can also use these options to edit and replace the weld points on faces when the original faces move due to product changes.

The **Reference Sheet** option previously available under **Settings**, is now available under **Face Sets**.

The command has the following enhancements:

- You can now locate the weld points using the **Projection Direction** options. These options are available when **Construction Method** is set to **Single** or **Multiple**, and **Reference Sheet** is set to **Top** or **Overlap**.

  You can also specify existing points in any plane, which can be projected along a specified vector or face normal to the reference sheet.



1 — Reference sheet

2 — Existing points

3 — Existing points projected onto the top/overlap sheet

- The **Connected Panels** option, available under **Settings** lets you specify the number of sheets to be welded. This option is available when **Construction Method** is set to **From Points**. You can use this option in conjunction with the **Run Connected Face Finder** option, to identify the correct number of faces for each weld point.

Where do I find it?

In the Modeling application, do one of the following:

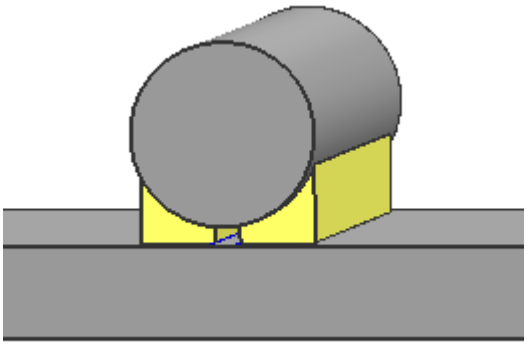
- Choose **Insert® Welding® Weld Point**.
- On the **Weld Assistant** toolbar, click **Weld Point** .

Groove Weld

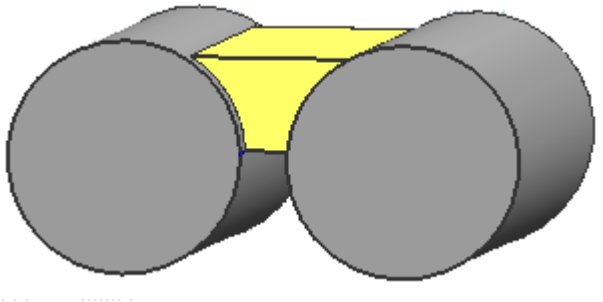
What is it?

The **Groove Weld** dialog box has new options under **Settings**:

- A **Use Fill In Construction** option is now available for the **Flared V Groove** and **Flared Bevel Groove** types of welds. If the selected faces do not touch, you can use this option to fill the approximate intersection location.

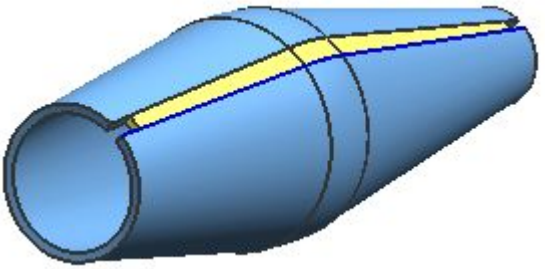


Flared Bevel Groove weld created using the Use Fill In Construction option



Flared V Groove weld created using the Use Fill In Construction option

- A **Single Face Set** option allows you to create a groove weld by selecting only a single face.



Butt Weld created using the Single Face Set option

- The **Allow Broken Link Bodies** option controls whether faces of linked bodies are available for selection.

- The **Allow Taper Angle** option enables you to show or hide the handles and dynamic input boxes for entering the start and end taper angle values.

The **Recreate Deleted Skip Welds** option, available under **Skip Weld Parameters** allows you to recreate an individual skip weld you deleted. This option is available only when you edit a skip weld.

Why should I use it?

You can use the **Groove Weld** enhancements to:

- Create flared groove welds when geometry does not touch.

- Create groove welds on a single face set. For example, you can create a groove weld on a cylinder.

- Select geometry from parts that contain broken links. This is useful in the edit mode, when you want to change faces from a body with broken links, to a new linked body.
- Hide the taper angle display until required.
- Recreate an individual skip weld you deleted.

Where do I find it?

In the Modeling application, do one of the following:

- Choose **Insert**® **Welding**® **Groove**.
- On the **Weld Assistant** toolbar, click **Groove** .

Feature Publish to Teamcenter

What is it?

Arc Weld feature Publishing

You can publish arc welds as objects in Teamcenter. You can publish all types of arc welds including:

- Groove Welds
- Fillet Welds
- Plug/Slot Welds
- Edge Welds

These welds are published to Teamcenter as PS Connection objects. The objects maintain knowledge of the parts they join.

Note

This functionality requires Teamcenter 2007.

Feature publishing of arc welds includes the following:

- Attributes from the individual weld features are attached to the Teamcenter object.

  NX attaches a feature form to each PS Connection object (PSOccurrence) in Teamcenter to capture the attributes of the arc weld object. This is the Teamcenter form object which contains the attribute title and its value.

- 3D visualization of a weld in Teamcenter's Portal Viewer is enabled.

Based on the geometry of the weld object, NX creates the JT file containing the associated 3D visualization data, and attaches the file to the PS Connection object. You can control the visibility of the published feature using functionality in Teamcenter.

- Connected parts information is passed to the Teamcenter object.

  The Connected Part information is used to establish a Connected_To relationship between the weld object and the item revision identified by the NX Connected Parts. In the PSE, select the weld object, and, from the shortcut menu select **connected_to** to identify the connected parts in Teamcenter.

- Feature data is synchronized between NX and Teamcenter.

  NX tracks features as they are created, modified, deleted or remain unchanged, and sends the updates to Teamcenter. This ensures that specific feature data is updated in Teamcenter without all the feature data being modified.

Support of Custom Spot Weld

Publishing custom spot welds to Teamcenter supports the display of the custom symbols in the Teamcenter window. To enable this functionality, create a JT file representing the custom symbol and place it in the folder named **$UGWELD_DIR\jt_files**. The format for the file name is *weld_xx.jt*, where *xx* corresponds to the point marker number. For example, for the 2T KPC Spot Weld, the name must be weld_44.jt. The point marker number is same as the marker number for the characteristics in Customer Defaults.

Why should I use it?

You can publish weld objects as individual PS Connection objects in Teamcenter so that other Teamcenter applications can use them.

Where do I find it?

In the Modeling application:

- Choose **Preferences® Teamcenter Integration**. In the **Teamcenter Integration** dialog box, click the **Feature** tab. Save the part or assembly which contains the weld objects.

  You can find the published individual weld objects under the weld item revision in Teamcenter **PSE**.

Mold and Die Tools

MPV Wall Thickness calculation

What is it?

When you use **Molded Part Validation** to calculate the thickness of any area in a solid model you can now:

- Save the results.
- Calculate more accurately with the **Ray** method.
- Choose 2–12 colors for the display of the results.
- Perform more accurate calculations on bigger and more complex models.
- Use the **Rolling Ball** method more effectively to calculate simple shapes, like cubes, cylinders and spheres.

Why should I use it?

These enhancements save time and make the Wall Thickness Checker more accurate. By saving the results, you can access them at any time without having to recalculate. Customizing the number of colors for displaying the wall thickness results gives you the flexibility of using the desired number of colors based on the requirements of your application.

Both methods of calculating wall thickness are improved. If you select the **Project to Face** check box, under **Ray Method**, the software projects the calculated points onto the face along the normal direction of the face for the points. This results in a more accurate result. Improvements to the **Rolling Ball** method improve the calculation speed on cubes, cylinders and other simple shapes.

Where do I find it?

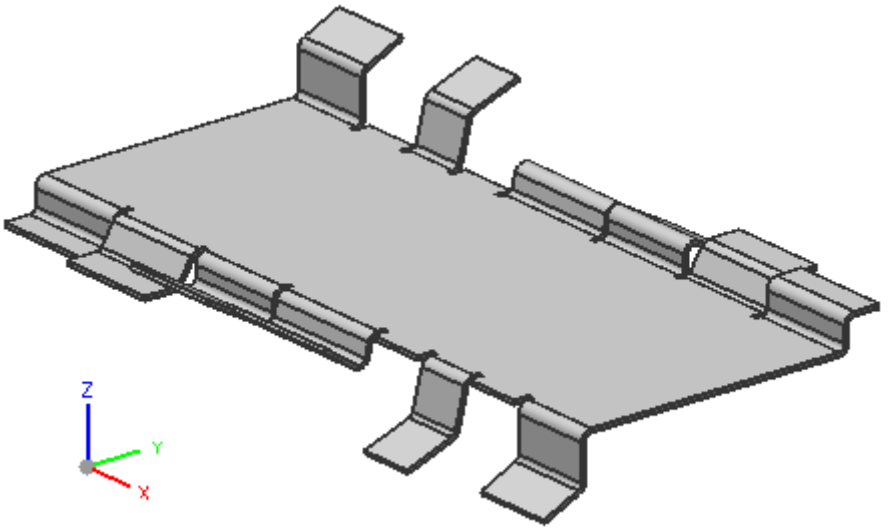
In the **Wall Thickness Check** dialog box:

- On the Calculate page , click **Save Results** . Select the **Project to Face** check box under **Ray Method** and select the
- On the Options page, from the **Number of Colors** list, select the number of colors you want used in the display of the calculation results.

Universal Z Bend

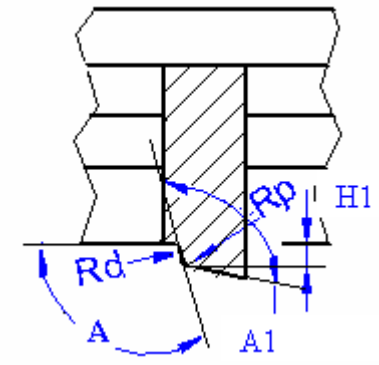
What is it?

You can now use **Universal Z Bend** to create Z and V bends that cover a greater range of bending inserts. When you load the insert and open the **Standard Parts Management** dialog box, the new catalog **UZ_Bending** exists for these inserts.



Several Universal Z Bends

These new bending inserts have two new parameters, **H1** and **A1**, to more precisely define the bends, bending punches and bending dies.

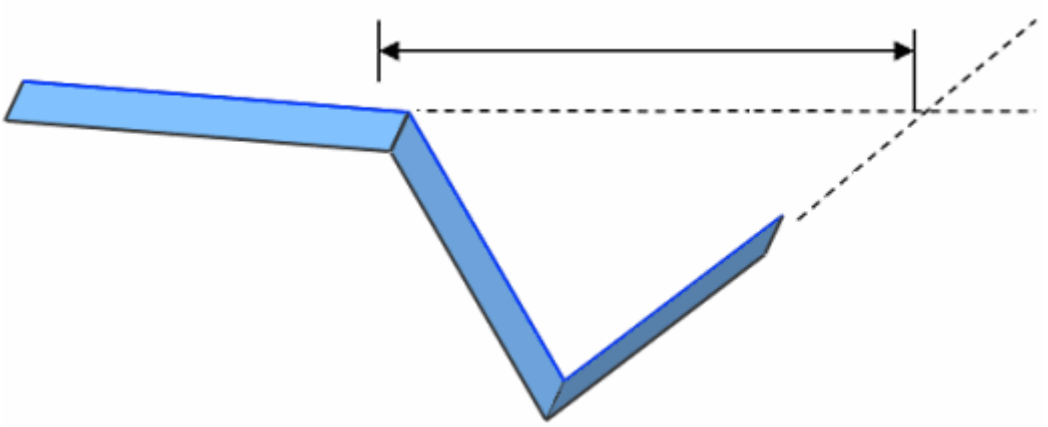


A diagram of a Universal Z Bend Down Punch.

.

Why should I use it?

The previous Z and V bends did not cover all possible bending situations. The **Universal Z Bend** allows for bends where the intersection of the two dashed lines below define the GW parameter.



The GW parameter of a Universal Z bend

Where do I find it?

1. On the **Progressive Die Wizard** toolbar, click **Insert Groups**.

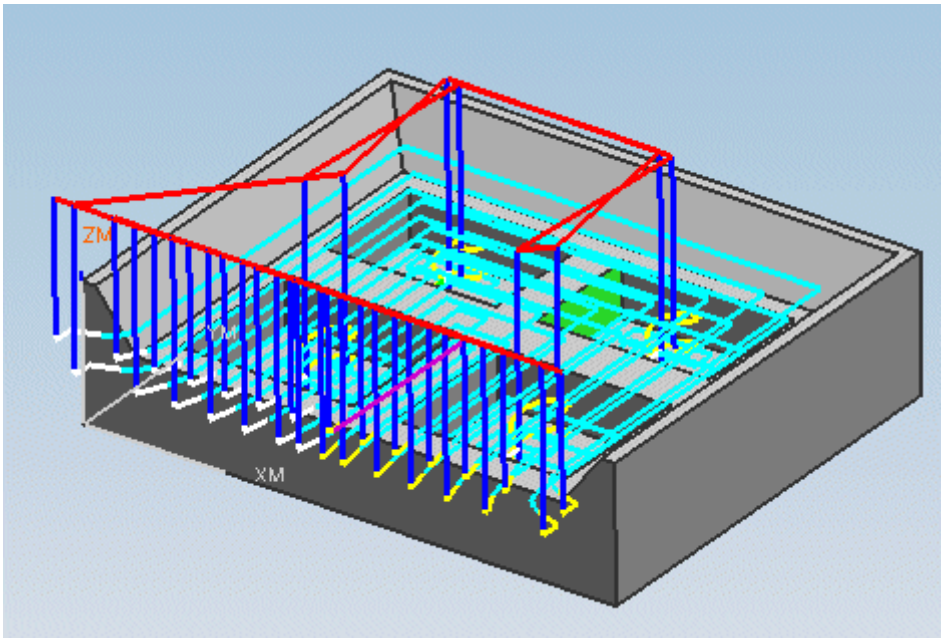
2. On the Bending page of the **Insert Group Design** dialog box, click **Universal Z Bend**.

Manufacturing

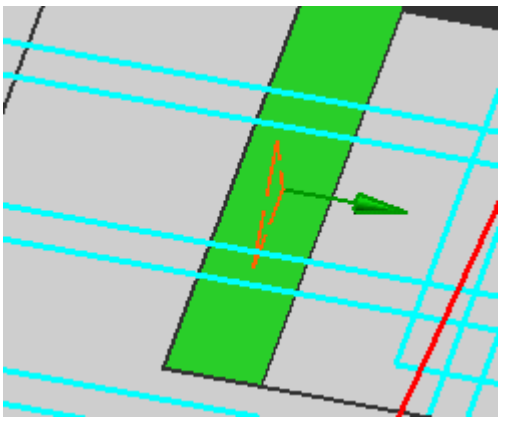
Tool path editing — trimming

What is it?

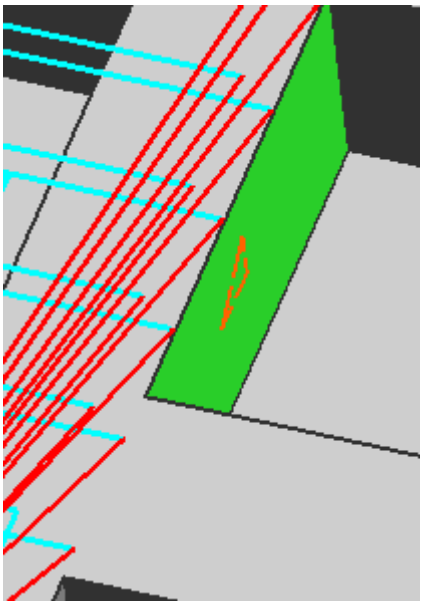
Tool path trimming is available for all milling operations. You can trim to a plane or curves as well as lines.



Original tool path



Trimming plane selected



Trimmed tool path

You have the following options for trimming:

- **Plane** lets you select any plane for the trimming plane.
- **Line in View** lets you specify a line to define a trimming plane normal to the current work plane, which is the rotated plane that you view.

  Tip

  Orient your work view before you trim, to get the trimming plane that you want.

- **Boundary** lets you define a boundary by selecting faces, curves or points.
- **Permanent boundary** lets you select a predefined permanent boundary. (This option was previously available.)

When you select the **Transfer Type** option **Clearance plane**, the trimming motions retract to your defined clearance plane, rapid along the clearance plane, then engage into the next cutting motion. You can define a local clearance plane for the trimmed tool path, or use the clearance plane defined for your operation (if any), which is the default.

Note:

- Edited tool paths are not associative, so there is no way to determine if they are out-of-date.
- Currently, you must turn on the **Path** column in the **Operation Navigator** to see whether the tool path was edited.

Why should I use it?

This is useful as a temporary fix to keep the cutter out of unwanted areas that are identified by shop personnel after the tool path is released for production.

Where do I find it?

Right–click the operation in the **Operation Navigator** and choose **Toolpath® Edit**.

Tool Path Divide

What is it?

Tool Path Divide is an option that lets you divide existing NC operations into one or more operations based on:

- Time of operation
- Length of operation

Why should I use it?

Tool Path Divide is used when:

- Cutting time is longer than the expected life of the cutter.
- The size of file may be too large for the machine tool controller memory.
- Specific requirements of a company dictate program size limits.

Where do I find it?

Right–click the operation in the **Operation Navigator**, Program Order view, and choose **Toolpath® Divide**.

Solid tool support in Teamcenter libraries

What is it?

You can now retrieve more tools stored in Teamcenter Resource Manager from the NX library. Teamcenter Resource Manager supports these tool types and attributes:

- User-modeled solid tools (new class in Teamcenter RM)
  - Probes
  - Generic (for example, a paint nozzle or welding gun)

Note

To use a solid tool, you must classify it as a solid tool in Teamcenter or NX will not load the part file.

- Milling form tools (new class in Teamcenter RM)
- Turning form tools (new class in Teamcenter RM)
- Tool holders

  There are additional holder attributes for the existing tool assembly and tool holder classes.
- Step drills
- Tracking points for milling and turning tools

Why should I use it?

This enhancement lets you access more library tools within a Teamcenter installation.

Where do I find it?

1. On the **Manufacturing Create** toolbar, click **Create Tool** .
2. In the **Create Tool** dialog box, click **Retrieve Tool from Library** .

Generic Motion

What is it?

Generic Motion now has new motion and control functions:

- The **Set Tracking Point** sub-operation lets you change tracking points on multi-point (star probe, multi-head paint gun) tools within the operation.

  For Parametric Milling and Drilling Tools:

  – Use the tool end center position for the tracking point.

  – You cannot change the tracking point on this type of tool.

  For Non-parametric Solid Tools (Standard and Probes):

  – The default tracking point is the first point in the Tracking Point list as defined in the tool.

  – Multiple tracking points are allowed in one operation.

  - The creation of new tracking points on the tool is not allowed.

  - The modification of X offset, Y offset, Z offset values is allowed.

  Changing the **Adjust Register** number is allowed for Parametric Milling and Drilling Tools and for Non-parametric Solid Tools.

- The **Rotate Tool** sub-operation lets you change the orientation of the tool in the spindle while in the operation. This is available only when the machine has the capability to orient the spindle under program control.

  Points to note are:

  - You must specify an absolute destination angle (**Destination Orient Angle**) to rotate the tool around the spindle. The original mounted position of the tool is at zero degree.

  - Rotation will be an absolute angle according to the right-hand rule of rotation.

  - Positive and negative values are allowed.

  - Once a **Rotate Tool** sub-operation rotates the tool, it remains active until changed by another **Rotate Tool** sub-operation or the current sub-operations ends.

  - At the end of the operation, any special orientation of the tool is not carried over to the next operation. Special orientations apply only to the operation in which they are programmed.

  - You must specify an orientation of zero degrees to return the tool orientation to its original mounted position.

Generic Motion also has the following enhancements:

- **Offset along a Face Normal** lets you position a tool at a point that is offset along the face normal from a point selected on that face.

- **Move along Vector Normal** lets you specify a point (instead of a distance), which will determine where the tool stops as it moves along the vector.

  The point can either be on the vector or the point will be projected perpendicular to the vector.

Why should I use it?

To give more options while positioning tools such as probes, welding heads, or paint spraying heads required for different applications.

Where do I find it?

1. In the **Create Operation** dialog box, select **mill_multi_axis** from the **Type** list.
2. In the **Operation Subtype** group, click **GENERIC_MOTION** .
3. In the **Sub-Operation** group, click **Add New Sub-Operation** .

   Set Tracking Point and **Rotate Tool** are in the **Type** list.

Integrated Simulation & Verification (ISV)

What is it?

The ISV Common Simulation Engine (CSE) now supports multi-channel machines and millturns. The merging lathe example in the mach kit *samples* directory runs in the CSE by default.

To use the CSE for multi-channel and millturn machines, uncomment the following line in the *machine.dat* file:

```
#CSE_FILES, ${UGII_CAM_LIBRARY_INSTALLED_MACHINES_DIR}sim_multi_channel_mm\cse_fi
```

Note

You must have the appropriate CSE files:

- Machine configuration file (*.MCF*)
- Controller configuration file (*.CCF*)
- Tool change program (*ToolChange.prg*)

Why should I use it?

This enhancement supports improved simulation accuracy.

Where do I find it?

See *Machine management in the Mach kit* in the Integrated Simulation & Verification (ISV) online Help for details on the ISV file directory structure.

Non Cutting Moves

What is it?

Non-cutting move enhancements let you override the initial engage and final retract. You can also control the initial approach and final departure.

The Engage page now has two new groups:

- **Initial Closed Area** controls the first engage into a closed area and has the same options as **Closed Area**. There is one additional option:
  - The default setting is **Same as Closed Area** which matches the **Closed Area** setting.
- **Initial Open Area** controls the first engage into an open area and has the same options as **Open Area**. There are two additional options:
  - **Same as Open Area** matches the **Open Area engage** setting.
  - **Same as Closed Area** matches the **Initial Closed Area** setting.

The Retract page now has one new group:

- **Final** controls the final retract and has the same options as **Retract**. There are two additional options:
  - **Same as Retract** matches the **Final** retract type to the **Retract** setting.
  - **Same as Engage** matches the **Final** retract type to the **Open Area** engage setting.

The Transfer/Rapid page now has one new group:

- **Initial and Final** controls the initial approach and final departure. The options are:
  - **Approach Type**
    - **Relative Plane** defines a plane that is the specified **Safe Clearance Distance** value along the tool axis above the initial engage point. The approach move travels directly from this plane to the engage point.
    - **Clearance** specifies an initial approach from the clearance plane.
    - **None**
  - **Departure Type**
    - **Relative Plane** defines a plane that is the specified **Safe Clearance Distance** value along the tool axis above the final retract point. The departure move travels directly to this plane.

- **Clearance** specifies a final departure to the clearance plane.
- **None**

Why should I use it?

These enhancements give you more control over the beginning and end of the tool path.

Where do I find it?

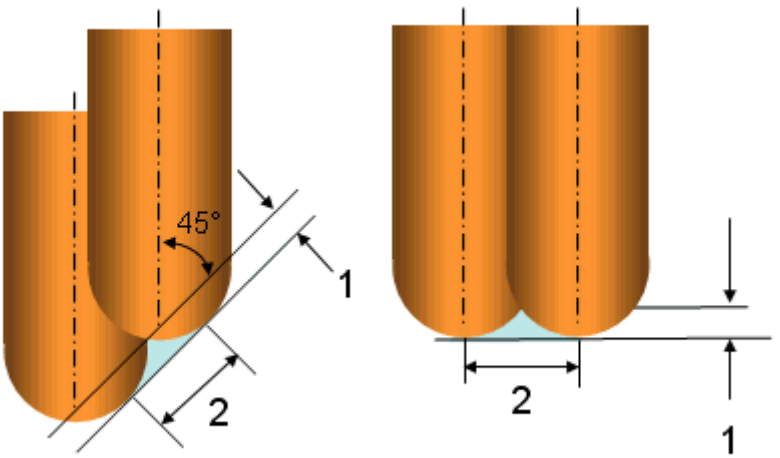
In a milling operation dialog box, in the **Path Settings** group, click **Non Cutting Moves** .

Streamline scallop stepover

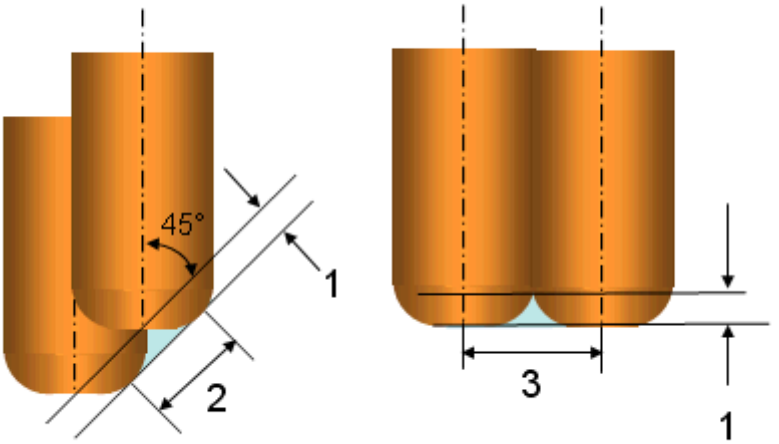
What is it?

Streamline has a new **Scallop** option for **Stepover** which specifies the distances between successive cut passes. You enter a **Scallop Height** value and the software calculates the stepover along a plane at a 45 degree angle to the tool axis.

Other drive methods use a plane perpendicular to the tool axis to calculate the stepover value. For equal **Scallop Height** values, **Streamline** will calculate the same stepover value for ball end tools and a smaller stepover value for other tool shapes, as shown below.



1 = scallop height, 2 = stepover distance



1 = scallop height, 2 = stepover distance calculated with Streamline, 3 = stepover distance calculated with other methods

Note

The software limits the stepover size to less than approximately two-thirds of the tool diameter regardless of the **Scallop Height** value entered.

Why should I use it?

The **Scallop** option limits scallop height to produce a smoother finish.

Where do I find it?

1. In the **Streamline** dialog box, in the **Method** group, click **Edit**.
2. In the **Streamline Drive Method** dialog box, expand the **Path Settings** group.

Automation

Knowledge Fusion

Product Template Studio

What is it?

Product Template Studio is a tool in NX that helps users to greatly simplify interaction with complex designs by allowing them to create a block-based user interface that serves as a guide for future interaction with a design. Once this interface is in place, other users can then choose to interact with

the design via the simplified interface, rather than interacting directly with the parametric feature history and/or model expressions.

Product Template Studio effectively modularizes a parametric design into an easily reusable template. The process of adding the block-based user interface is completely interactive, and involves no coding at all.

Design validation can be built into the templates through the inclusion of Requirements Checks. Documentation for templates can also be easily attached and referenced via the simplified template user interface.

Once packaged for modularity, templates can then also be snapped together to form larger systems of templates.

Product Template Studio consists of two new NX licenses:

- Product Template Studio Author — Authoring is conducted in an external graphical Java application. It is in this environment that you can create the new user interface for an NX part or assembly, associate documentation with the template, and predefine systems of combined templates. You can then use templates saved in this authoring environment inside NX using only a Product Template Studio Consumer license.

- Product Template Studio Consumer — Allows a user to see and interact with Product template Studio interfaces from within the NX application. Parts or assemblies containing a Product Template Interface will display a small light bulb icon in the Assembly Navigator. A new MB3 action in the Assembly Navigator or in the graphics area (Edit Reusable Component…) allows easy access to the Product Template interface.

  You can also store Templates in the new **Reuse Library**. The **Reuse Library** must be enabled in the Customer Defaults. Under Gateway, select **Reuse Library**, and on the General page, select **Display Reuse Library**. When Templates are dropped from the Reuse Library into a design, a copy is automatically created of the template part or assembly, and it is automatically added to the current Work Part as a component (or sub-assembly.)

Why should I use it?

Product Template Studio greatly simplifies interaction with complex designs, and alows the creator of a model to package the design for reuse by other users.

Where do I find it?

The Product Template Studio Author environment can be found in the **NX Tools** folder in the standard NX installation (in the Windows Start menu on the Windows platform.)

Parts or assemblies already containing a Product Template Interface display a small light bulb icon in the Assembly Navigator. A new MB3 action in the Assembly Navigator or in the graphics area (**Edit Reusable Component**...) allows easy access to the Product Template interface.

NXOpen API

API for Make Sketch Internal

What is it?

The **Make Sketch Internal** and **Make Sketch External** options that you can get by right-clicking supported features in the **Part Navigator** are now available as API functions:

```
MakeSketchExternal( )

MakeSketchInternal( )
```

You can use these new API functions on any feature that supports internal sketches (such as **Extrude** and **Revolve**).

Where do I find it?

These new functions are available in NX Open for C/C++, Java, and .Net.

Translators

NX to JT

Section view support

What is it?

You can now write the section views present in an NX part or assembly to JT files using the new **doSectionViews** configuration option. Section views are model views with cut geometry, hence section views in JT files appear as model views in the PMI tree within Teamcenter Visualization.

Set **doSectionViews** to true, to write the following additional information to JT:

- Section cut geometry
- PMIs present in section views

Note

- Section views functionality is supported only in **File® Export® JT** and "-honour_structure" command line modes of translation.
- Section views are not supported in the following cases:
  - Managed or "-generate_assy_jt" mode of translation.
  - If you set mergeSolids = false in the configuration file.
- Due to the addition of section cut geometry information, the overall JT file size may increase.

New configuration option for section view support

What is it?

A new option, **doSectionViews**, is added to the *tessUG.config* file. When set to true, this option writes section views in an NX part or assembly, to JT. The following new information is added:

- Section cut geometry

  Note

  Due to the addition of section cut geometry, the overall JT file size may increase.

- PMIs present in section views

By default, this option is set to false.

Why should I use it?

Use the **doSectionViews** configuration option to control the writing of section views in NX parts or assemblies to JT.

Where do I find it?

This configuration option is located in the UGconfig section of the *tessUG.config* file.

Configuration options

In NX 5.0.1, to better translate NX enities to JT, some new configuration options are added, some are enhanced, and some are discontinued.

New configuration options

What is it?

The following options are added to the *tessUG.config* file.

Option	Description	Value
activateGDTPMI	Writes the following Geometric Dimensioning and Tolerancing (GD&T) symbols to JT: • Datum feature symbol • Datum target • Feature control frame **Note** This option replaces the **activateFcfPMI** and **activateDatumPMI** options, which are no longer valid.	true false
activateSymbolPMI	Writes the following GD&T symbols to JT: • Locator designator • Surface finish • Line weld • Custom symbol • User-defined symbol	true false
advancedMaterials	Writes the advanced materials and lights information to JT. When set to **true**, the following additional information is added: • Materials applied to NX parts • Texture images applied to NX parts • Lights defined in the NX part **Note**	true false

	You must have the NX STUDIO license to access this functionality.	
multiCADJT	Writes the NX entities to version 9.0 JT files. These are smaller in size and contain additional data recommended for viewing, and multi-CAD workflows and supplier exchange. **Note** If your application cannot read version 9.0 JT files, set **multiCADJT** to **false**.	true false
getCADProperties	Writes specific NX properties to the JT file. For example, if you set the option as follows: getCADProperties = "CAD_DENSITY,CAD_MASS, CAD_MATERIAL", then the specified NX properties, that is, density value, mass value and material name, are written to the JT file.	<string> You can set some or all of the following values: • CAD _DENSITY — Displays the density value. • CAD _MASS — Displays the mass value. • CAD _MATERIAL — Displays the material name. • CAD _VOLUME — Displays the volume value. • CAD_SURFACE _AREA — Displays the surface area value.

		• CAD_YOUNGS_MODULUS — Displays the Youngs Modulus value. • NONE — Displays none of the above values. • ALL — Displays all of the above values.

Where do I find it?

These configuration options are located in the ugConfig section of the *tessUG.config* file.

Updated configuration options

What is it?

The following configuration options are enhanced to better translate NX entities to JT.

Option	Description	Value
activateNotePMI	Writes all notes present under **PMI® Note** and **PMI® Notes**, to JT.	true false
activateCsysPMI	Writes the following NX entities to JT: • Datum plane • Datum axis • Datum CSYS • Datum point	true false
activateDimPMI	Writes all dimension entities present under **PMI® Dimension** to JT.	true false

Where do I find it?

These options are located in the JT configuration file named tessUG.config.

Discontinued configuration options

What is it?

The following configuration options are no longer available:

- **activateFcfPMI** — This is replaced by the activateGDTPMI option.
- **getCADVolume** — This is replaced by the getCADProperties option.
- **activateDatumPMI** — This is replaced by the activateGDTPMI option.
- **getCADDensity** — CAD density is translated by default. You can also control this using the getCADProperties option.
- **getCADSurfaceArea** — This is replaced by the getCADProperties option.
- **newPMI** — This option is no longer valid and is removed.

Writing V9.0 JT files for multi-CAD workflows

What is it?

NX is now capable of writing JT files to the latest V9.0 JT format. These files include the Topomesh format that compresses the tessellation (facet) part of the file size by approximately 50 percent, and requires around 15 percent less viewer memory. These files do not include Libra JT.

To write to the V9.0 JT format, set **multiCADJT** to **true** in your configuration file.

Why should I use it?

The enhanced format and additional content is particularly recommended for multi-CAD workflows and supplier exchange.

Note

- NX 5.0 and NX-Ideas 5.0 and later versions can use version 9.0 JT files.
- You must install Teamcenter 2007.1 or Teamcenter Visualization 2007.1 to view version 9.0 JT files.
- If you write JT files with the **multiCADJT** option set to **true**, and then want to read those files with an earlier version viewer, Teamcenter Visualization 2007.1 will export to an earlier JT format.

Texture, material and light support

What is it?

You can now write the results of basic Studio display mode rendering to your JT files, ready for display in Teamcenter. When you set the **advancedMaterials** option to **true**, the following additional information is written to JT:

- Materials assigned to tessellated geometry
- Texture assigned to tessellated geometry
- Lighting characteristics for a given view in NX

Note

- You must have the NX STUDIO license to access this functionality.
- The quality of the display of JT files containing advanced materials in the Teamcenter Visualization suite of products is similar to the quality of the display in NX basic Studio.
- Texture information is assigned to the tessellated representation. If the user performs re-tessellation in Teamcenter Visualization, newly generated levels of detail (LODs) do not have texture information.
- The Moldtech set of NX Textures are proprietary to LightWork Design Ltd., and are not currently written to JT.

Weld support

What is it?

You can now write custom spot weld symbols to JT. The existing incremental counter value appended to the spot weld name and displayed in the PMI tree of Teamcenter Visualization, is now replaced with the NX attribute ID of the weld feature.

Note

You need Teamcenter Visualization 2007.1 to view the symbol colors in Teamcenter Visualization.