Chapter

1 What's New in NX 7.5.3

Chapter

2 Gateway

Real time shadows for Advanced Studio views

Advanced Studio Ambient Shadows

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

Shadows rendered considering the ambient lighting

Shadows rendered using a single light source

When you use the command, you can:

- Create new visualization scenes with different Advanced Studio Ambient Shadows settings.
- Edit the **Advanced Studio Ambient Shadows** settings in the **Basic Scene Editor** dialog box to produce optimal results for the part. The results are displayed dynamically. The settings are saved with the part.
- Delay the update of the shadows by using the **Disable Dynamic Update Advanced Studio Ambient Shadows** Visualization preference.

Note

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

Where do I find it?

Prerequisite	**Advanced Studio Ambient Shadows** is supported only on devices that support the **Advanced Studio Display** mode and requires availability of DirectX 10. The **Advanced Studio Display** mode must be enabled. On the **Visualize Shape** toolbar, click **Advanced Studio Display**  , or choose **View**® **Visualization**® **Advanced Studio Display** to enable the **Advanced Studio Display** mode.

Advanced Studio Ambient Shadows command

Toolbar	**Visualize Shape**® **Advanced Studio Ambient Shadows** 
Location in dialog box	**Shadows** tab® **Advanced Studio Ambient Shadows (Part Settings)** group® **Ambient Shadows** check box

Ambient Shadows check box

Toolbar	**Visualize Shape**® **Basic Scene Editor** 
Location in dialog box	**Shadows** tab® **Advanced Studio Ambient Shadows (Part Settings)** group® **Ambient Shadows** check box

Advanced Studio Ambient Shadows settings

What is it?

Advanced Studio Ambient Shadows settings are specific to a part and are applicable to all rotatable advanced studio views in the part layout. These settings are saved with the part.

Advanced Studio Ambient Shadows (Part Settings)

Appears when **Advanced Studio Display** is turned on.

Ambient Shadows Lets you display ambient shadows.

Quality Controls the quality levels. High settings impact the performance.

High

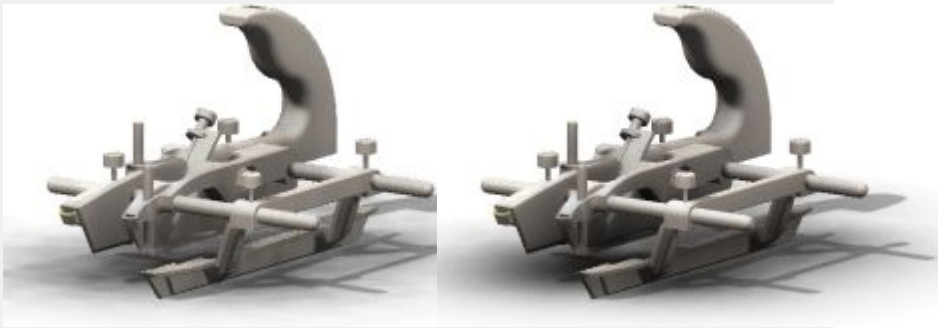
Uses a full-sized buffer with higher resolution sampling.

Medium

Uses a half-size buffer with finer sampling.

Low

Uses a half-size buffer with coarse sampling.



Low quality level **High quality level**

Contrast

Lightens or darkens the ambient occlusion effect.

Low contrast

High contrast

Radius

Controls the maximum distance from a point to be considered for occlusion. As this value defines the physical size of the effect, it is dependent on the model.

Low radius

High radius

Blur Radius	Controls the radius of blurring of the ambient shadows effect from the point of calculation. This value is measured in pixels. Set smaller values for sharper details.

Low blur radius

High blur radius

Where do I find it?

Prerequisite	The **Advanced Studio Display** mode must be enabled.
Toolbar	**Visualize Shape® Basic Scene Editor** 
Location in dialog box	**Shadows** tab® **Advanced Studio Ambient Shadows (Part Settings)** group

Disabling dynamic update for ambient shadows

What is it?

Use the **Disable Dynamic Update Advanced Studio Ambient Shadows** preference to delay the update of ambient shadows that are based on the ambient lighting of the entire scene.

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

Where do I find it?

Menu	**Preferences® Visualization Performance**
Location in dialog box	**Visualization Performance Preferences® General Graphics** tab® **Studio Views** group® **Disable Dynamic Update Advanced Studio Ambient Shadows** check box

Chapter

3 Modeling

Part repair for migrated Pro-E files

What is it?

You can now perform part repair on parts previously migrated using Content Migration Manager for Pro/ENGINEER (PTC).

After you have the migrated part file, you can perform fidelity checks on it in NX and manually correct failed or unsupported features in the list generated by the checks.

Why should I use it?

You can verify the accuracy of migrated Pro/ENGINEER parts from within NX.

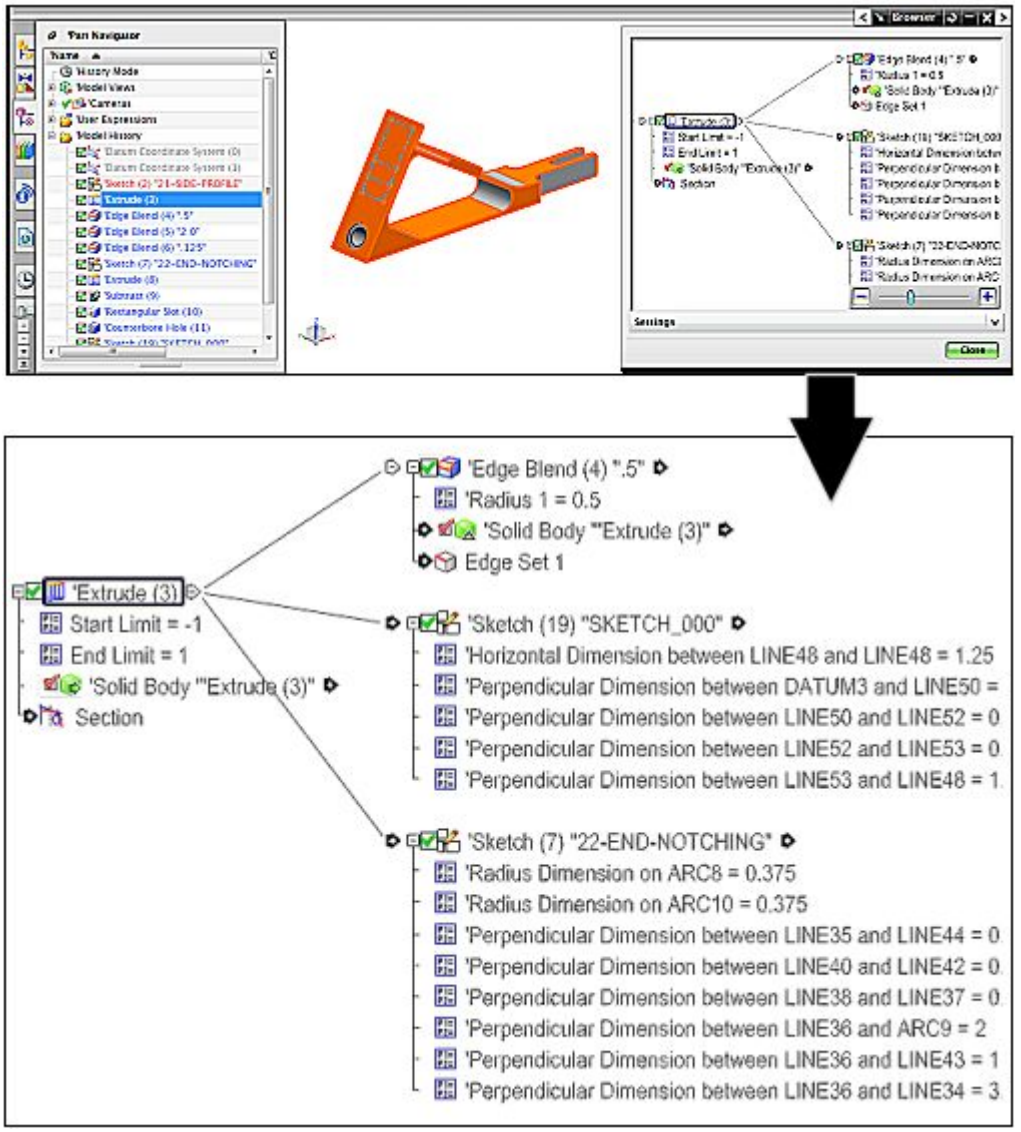
Where do I find it?

Prerequisite	The part file you use must have been created using Content Migration Manager for Pro/ENGINEER
Menu	**File**® **Properties**
Location in dialog box	**Pro/Engineer Migrated** tab® **Check Feature Fidelity**

New feature and relations Browser

What is it?

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



The following types of features appear in the Feature Browser:

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

You can interact with nodes in the browser display.

- To expand a node click the node.

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

You can also browse non-feature relations.

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

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

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

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

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

When you click a node in the browser, the respective object is highlighted in the graphics window and in the **Part Navigator**, either as it exists now or as it was when it was created.

Why should I use it?

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

Where do I find it?

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

Part Navigator	Right-click a feature® **Browse** Right-click an expression® **Browse**

Feature timestamp sequencing

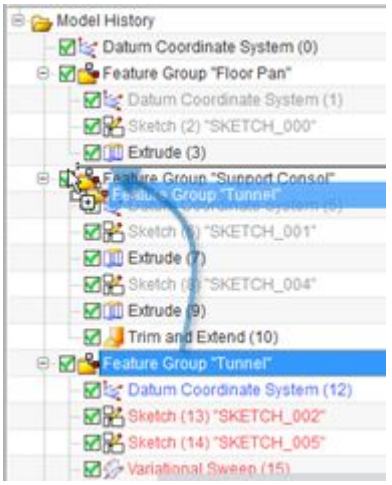
What is it?

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

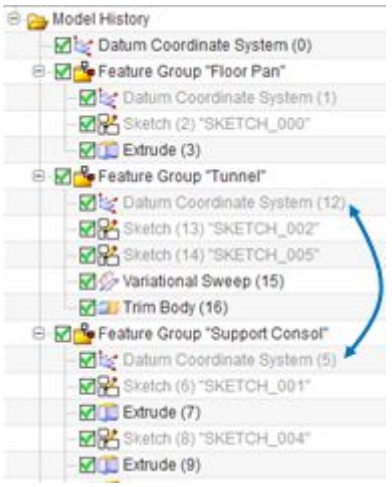
- When you create or reorder feature groups, the member feature timestamps are re-sequenced.

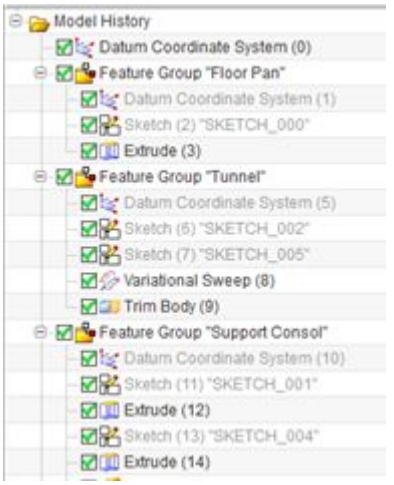
- When you create or reorder feature groups, the gaps are removed in the feature timestamp sequence are removed.

You cannot re-sequence features of some parent child relationships.



Reorder feature group





Previous result

Updated result

Why should I use it?

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

Where do I find it?

Application	Modeling and Shape Studio

Group commands

What is it?

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

- You can now create sketch groups when working in Drafting.
- When a sketch is in Modeling and in Drafting, you can select curves and dimensions and group them using the shortcut menu.

- In the **Part Navigator**, in the Drafting application, sketch groups now appear under the sketch node in the sheet.
- Group labels and their function are now consistent across applications:
  - **New Group** — Creates a new regular group
  - **New Sketch Group** — Creates a sketch group
  - **New Active Sketch Group** — Creates a new empty active sketch group

Where do I find it?

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

Durability analysis in the Modeling application

What is it?

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

Why should I use it?

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

Where do I find it?

Durability Wizard

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

Stress Wizard

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

Chapter

4 Assemblies

Synchronize Links

What is it?

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

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

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

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

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

Why should I use it?

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

Where do I find it?

Menu	**Assemblies® WAVE® Synchronize Links**

Chapter

5 Advanced Simulation

Solver version support

For each released version of NX, the following tables list the supported solver versions for import, export, and the post-processing of results. Note:

- The version listed in the “Import ASCII” and “Import Binary” rows is the solver version that was generally available when the NX version was released. In general, the import of the solver ASCII and binary files should be upwards compatible. Therefore, you should be able to import them into the most recent version of NX. However, in general:
  - ASCII files are backwards compatible for import into NX. If you import an ASCII file from a newer version of the solver than is officially supported, the software simply ignores any new fields/options that aren’t supported in the current NX release.
  - Binary files are not backwards compatible. For example, you can import a binary file created by NX Nastran 5.0 into NX 6.0.2 , but you might not be able to import a binary file created by NX Nastran 6.1 into NX 5.
- The version listed in the “Export ASCII” rows is the solver version that was available when the NX version was tested. In general, the exported solver input file is upwards compatible for that solver. Backwards compatibility is not guaranteed. For NX Nastran, the **Model Setup Check** function in Advanced Simulation tries to flag potential version incompatibility issues.
- The version listed in the “Post-processing Results” rows is the version of the solver results that was tested in the listed NX version. In general, results from earlier solver versions are also supported.

NX7 releases

Solver	File Type	NX 7	NX 7.5	NX 7.5.1	NX 7.5.2	NX 7.5.3
NX Nastran	Import ASCII (.dat)	6.1	7.0	7.0	7.1	7.1
	Import Binary (.op2)	6.1	7.0	7.0	7.1	7.1
	Export ASCII (.dat)	6.1	7.0	7.0	7.1	7.1
	Post-processing of Results	6.1	7.0	7.1	7.1	7.1
MSC Nastran	Import ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010
	Import Binary (.op2)	2008r1	2008r1	2008r1	2008r1	2010
	Export ASCII (.dat)	2008r1	2008r1	2008r1	2008r1	2010
	Post-processing of Results	2008r1	2008r1	2008r1	2008r1	2010
Abaqus	Import ASCII (.inp)	6.8-1	6.9–1	6.9–1	6.9-1	6.10
	Import Binary	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9	6.9	6.9	6.10
	Post-processing of Results (.fil)	6.8-EF2	6.9.2	6.9.2	6.10-1	6.10-1
	Post-processing of Results (.odb)	6.8-EF2	6.9-EF1	6.9-EF2	6.9-EF2	6.10-EF1
ANSYS	Import ASCII (PREP7, CDB)	12	12.1	12.1	12.1	13
	Import Binary (.rst, .rth)	12	12.1	12.1	12.1	13
	Export ASCII (.inp)	12	12.1	12.1	12.1	13
	Post-processing of Results	12	12.1	12.1	12.1	13
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A	971R3.2.1	971R3.2.1	971R3.2.1

NX 6 releases

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3	NX 6.0.4	NX 6.0.5
NX Nastran	Import ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Import Binary (.op2)	6.0	6.1	6.1	6.1	6.1	7.0
	Export ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Post-processing of Results	6.0	6.0	6.1	6.1	7.0	7.0
MSC Nastran	Import ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Import Binary (.op2)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Post-processing of Results	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
Abaqus	Import ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Post-processing of Results (.fil)	6.7-5	6.8-1	6.8-3	6.8-EF2	6.8-EF2	6.8-EF2
	Post-processing of Results (.odb)	N/A	N/A	N/A	6.8-EF	6.8-EF2	6.9-EF2
ANSYS	Import ASCII (PREP7, CDB)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Import Binary (.rst, .rth)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Export ASCII (.inp)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Post-processing of Results	11 SP1	11 SP1	11 SP1	11 SP1	12.0	12.1

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3	NX 6.0.4	NX 6.0.5
LS-DYNA	Import ASCII	N/A	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R2	971R2	971R3.2.1	971R3.2.1	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A	N/A	N/A	N/A	N/A

NX 5 releases

Solver	File Type	NX 5	NX 5.0.1	NX 5.0.2	NX 5.0.3	NX 5.0.4	NX 5.0.5	NX 5.0.6
NX Nastran	Import ASCII (.dat)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Import Binary (.op2)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Export ASCII (.dat)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Post-processing of Results	5.0	5.0	5.1	5.1	5.1	5.1	6.0
MSC Nastran	Import ASCII (.dat)	2005	2005	2007	2007	2007	2007	2007r1
	Import Binary (.op2)	2005	2005	2007	2007	2007	2007	2007r1
	Export ASCII (.dat)	2005	2005	2007	2007	2007	2007	2007r1
	Post-processing of Results	2005	2005	2007	2007	2007	2007	2008r1
Abaqus	Import ASCII (.inp)	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.7-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.7-1
	Post-processing of Results	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.8-1

Solver	File Type	NX 5	NX 5.0.1	NX 5.0.2	NX 5.0.3	NX 5.0.4	NX 5.0.5	NX 5.0.6
ANSYS	Import ASCII (PREP7, CDB)	10	10	11	11	11	11	11
	Import Binary (.rst, .rth)	10	10	11	11	11	11	11
	Export ASCII (.inp)	10	10	11	11	11	11	11
	Post-processing of Results	10	11	11	11	11	11	11 SP1

NX 4 releases

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

Solver	File Type	NX 4	NX 4.0.1	NX 4.0.2	NX 4.0.3	NX 4.0.4
ANSYS	Import ASCII (PREP7, CDB)	8	9	9	10	10
	Import Binary (.rst, .rth)	8	9	9	10	10
	Export ASCII (.inp)	8	9	9	10	10
	Post-processing of Results	9	9	9	10	10

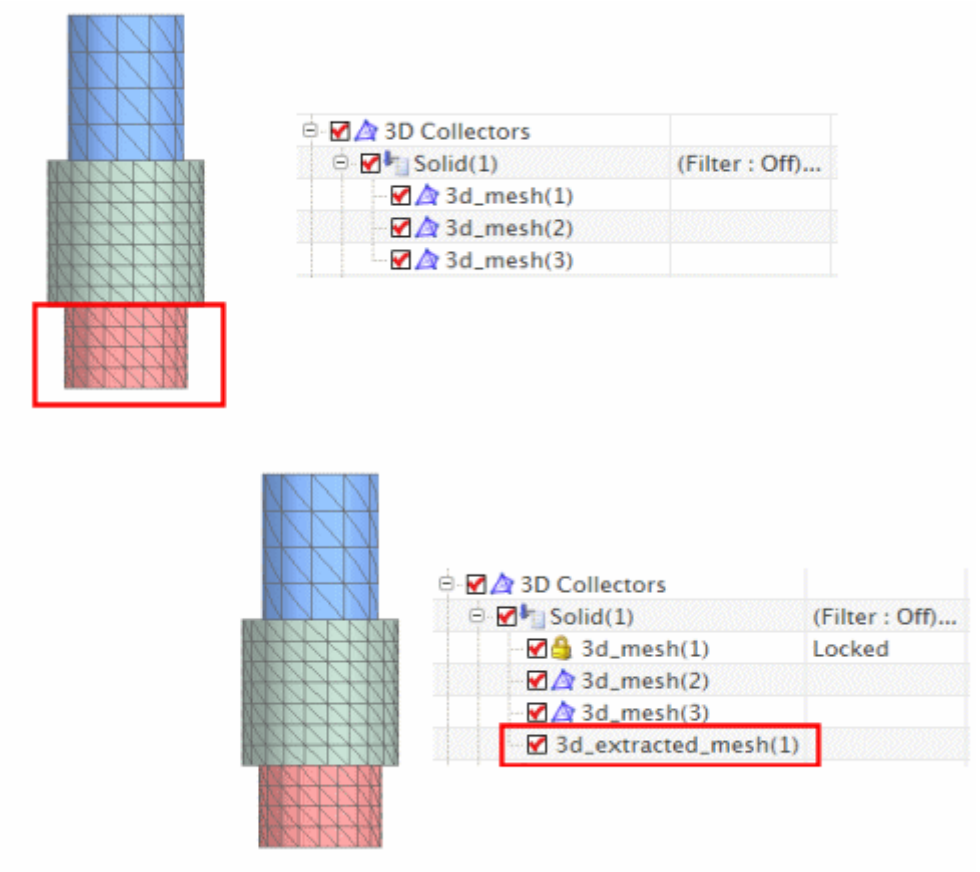
Element Extract enhancements

What is it?

The **Element Extract** command allows you move selected elements out of a source mesh into a new mesh.

Beginning with this release, the extracted mesh retains the mesh associated data and display properties of the source mesh. In previous releases, the extracted mesh used default settings for mesh associated data and display properties.

The following graphic shows the meshes listed in the **Simulation Navigator** before and after using **Element Extract**. The source mesh includes all of the pink elements in 3d_mesh(1). The new mesh is 3d_extracted_mesh(1).



Where do I find it?

Application	Advanced Simulation
Prerequisite	A FEM file with meshes displayed.
Toolbar	**Element Operations** toolbar® **Element Extract** 
Menu	**Edit® Element® Extract**

Merge Meshes enhancements

What is it?

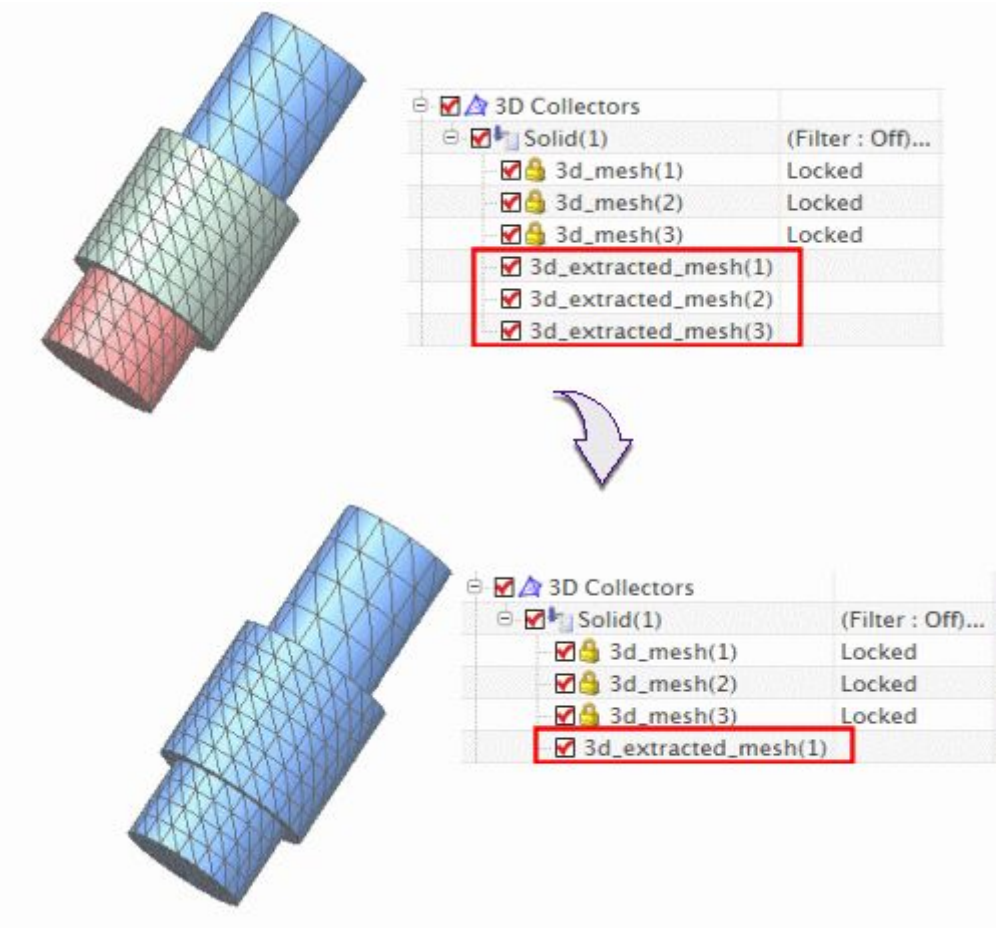
The **Merge Meshes** command lets you merge manually created, extracted, or imported meshes. The meshes to be merged must not be associated with any geometry, and must be located in the same mesh collector.

Beginning with this release, when you use this command, the first mesh that you select becomes the merged mesh. The software then adds the remaining selected meshes to the first mesh. The merged mesh retains characteristics of

the first mesh, including the mesh name, mesh associated data, and mesh display settings.

In previous releases, merged meshes were placed in a new **Simulation Navigator** node that had a unique name. The merged mesh display used default settings.

The following graphic shows the mesh names and display colors before and after three extracted meshes are merged.



Where do I find it?

Application	Advanced Simulation
Prerequisite	Meshes that are not associated with geometry and are located in the same mesh collector.
Toolbar	Select mutiple meshes® right-click a selected mesh® **Merge Meshes**

Improvements to Nastran CFAST/CWELD connections

What is it?

This release includes some improvements to the **CFAST/CWELD Connection** connection command that was introduced in the NX 7.5.2 release.

- You can now use the **CFAST/CWELD Connection** command if you are working in either the NX Nastran or MSC Nastran solver environments. In NX 7.5.2, the **CFAST/CWELD Connection** was only supported in the NX Nastran environment.

- There is now better correspondence between the connections that the **CFAST/CWELD Connection** command creates and Nastran's requirements. In NX 7.5.3, the **CFAST/CWELD Connection** command performs additional checks that help ensure that the connections you create will solve in Nastran.

Where do I find it?

Application	Advanced Simulation
Prerequisite	An active FEM file with Nastran as the specified solver
Toolbar	**Element Operations** toolbar® **CFAST/CWELD Connection** 
Menu	**Insert**® **Mesh**® **CFAST/CWELD Connection**

Transverse temperature gradient mapping to structural solutions

What is it?

The mapping solver can now write transverse temperature gradient results to the Nastran, Abaqus, and ANSYS input files.

- In addition to the TEMP card, the Nastran input file now contains the TEMPP1 card. For every transverse gradient element, the TEMPP1 card contains the element's temperature, which is mapped to the element's center of gravity, the elemental linear thermal gradient, and the top elemental and bottom elemental temperature.

- For every transverse gradient target node, the Abaqus input file contains the nodal temperature and the nodal linear thermal gradient.

- In addition to the BF card, the ANSYS input file now contains the BFE card. For every transverse gradient element, the BFE card contains the top and bottom elemental temperature for all N nodes on the element.

Note

The transverse gradient target nodes are all the nodes selected from the **Destination Nodes** group of the **Transverse Gradient Target Set** dialog box and all the nodes connected to the elements selected from the **Destination Elements** group.

The transverse gradient target elements are all the elements selected for the **Destination Elements** group.

Why should I use it?

You can now easily map transverse temperature gradients to Nastran, Abaqus, or ANSYS solutions for use in structural analyses, such as thermal contact modeling and thermo-elastic distortion analysis.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Coupled Thermal-Flow	Advanced Thermal/Flow with ESC
	Mapping	Thermal-Flow
NX Space Systems Thermal	Thermal	Space Systems Thermal
	Mapping	Thermal
NX Thermal and Flow	Thermal	Thermal
	Coupled Thermal-Flow	Advanced Thermal Thermal-Flow
	Mapping	Advanced Thermal-Flow Thermal-Flow

Where do I find it?

Defining a transverse gradient pair in the source model

Application	Advanced Simulation
Prerequisite	A solution with **Analysis Type** set to **Thermal** or **Coupled Thermal-Flow**.
Toolbar	**Advanced Simulation** ® **Mapping** 
Simulation Navigator	Right-click the **Constraint Set** container node ® **New Constraint** ® **Mapping**
Location in dialog box	**Type** ® **Transverse Gradient Pair**

Defining a transverse gradient target set in the target model

Application	Advanced Simulation
Prerequisite	A solution with **Analysis Type** set to **Mapping**.
Toolbar	**Advanced Simulation** ® **Transverse Gradient Target Set** 
Simulation Navigator	Right-click the **Constraint Set** container node ® **New Constraint** ® **Transverse Gradient Target Set**

Relaxation of mapped temperature bounds

What is it?

When you map the temperatures of the source model onto the target model, you can now relax the extrapolation limits for mapped temperatures.

The maximum and minimum temperatures in the target model can now be limited by the following bounds:

$$\text{Upper limit} = T_{max} + percentage \cdot \text{abs}(T_{max} - T_{min})$$

$$\text{Lower limit} = T_{min} - percentage \cdot \text{abs}(T_{max} - T_{min})$$

where T_{max} and T_{min} are the bounds of the entire source model. The *percentage* is the relaxation of the mapped temperature limits that you specify in the **Relaxation of Mapped Temperature Limits, % (Tmax–Tmin)** box.

Why should I use it?

When there are target nodes outside the geometric bounds of the source model, you can relax the temperature limits to help extrapolate temperatures to these target nodes.

Supported solvers and analysis types

Solver	Analysis Type	Solution Type
NX Electronic Systems Cooling	Mapping	Thermal-Flow
NX Space Systems Thermal	Mapping	Thermal
NX Thermal and Flow	Mapping	Thermal-Flow

Where do I find it?

Application	Advanced Simulation
Toolbar	**Advanced Simulation** toolbar ® **Solution**  **Advanced Simulation** toolbar ® **Solve**  ® **Edit Solution Attributes**
Menu	**Insert** ® **Solution** **Analysis** ® **Solve** ® **Edit Solution Attributes**
Simulation Navigator	Right-click the Simulation file node ® **New Solution** Right-click the solution node ® **Edit**
Location in dialog box	**Mapping Details** tab ® **Settings** group

Durability analysis in the Modeling application

What is it?

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

Why should I use it?

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

Where do I find it?

Durability Wizard

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

Stress Wizard

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

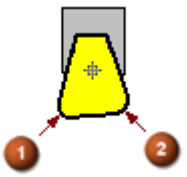
Chapter

6 Manufacturing

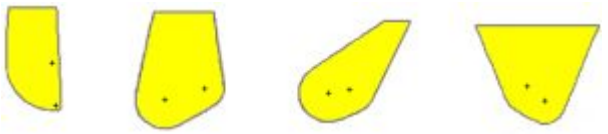
Non-Identical radii for grooving

What is it?

You can now specify two different size radii on the cutting edge of a user defined insert shape on a grooving tool. The grooving turning processors will correctly detect cut regions according to the custom user defined insert shape and machine the cut region more accurately than before.



Examples of grooving tool insert shapes that can be defined with two non-identical radii.



Why should I use it?

Use this functionality only when you want to machine a groove that has two different size radii more efficiently.

Where do I find it?

Application	Manufacturing
Prerequisite	When creating the tool, you must select a groove tool subtype.
Menu	**Insert® Tool**

Location in dialog box	**Type** group® **turning**® **Tool Subtype** group® **OD_GROOVE_L**  , **FACE_GROOVE_L**  , or **ID_GROOVE_L**  ® **Grooving Tool Standard** dialog box® **Tool** tab® **Insert** group® **Insert Shape**® **User Defined**

Chapter

7 CMM Inspection Programming

General enhancements

Link to PMI enhancements

What is it?

When you link to PMI (Product and Manufacturing Information) data, an information window reports on all Geometric Dimensioning and Tolerancing (GD&T) and dimensional PMI markup linked to Inspection Programming.

In the **Inspection Navigator**:

- Items marked as successfully linked appear in their appropriate feature, tolerance, and path locations.
- PMI markup not supported or used by Inspection Programming, such as notes and labels, do not appear.
- Items marked as linked but incomplete also display the ⊘ icon. These may require some manual modification, such as clearly linking the markup to new or existing inspection features, to reestablish links between markup and inspection features.

Why should I use it?

The PMI report lets you quickly confirm that your PMI markup has successfully linked to Inspection Programming. In **Inspection Navigator**, you can compare the results with the new inspection features and tolerances and make changes in Inspection Programming, PMI, or both.

Where do I find it?

Application	Inspection Programming
Prerequisite	You must define feature-based tolerance annotations in *PMI* and create an inspection file.
Toolbar	**Operations® Link to PMI** 
Menu	**Inspection Navigator® Operation® Link to PMI**

Inspection Navigator enhancements

What is it?

There are many enhancements to the **Program Order** view of the **Inspection Navigator**.

- You can reorder inspection paths and other elements.
  - You can click the element to be moved and then drop it on the element that it should follow.
  - You can select multiple elements by pressing Ctrl, and then drag them to a new location. If you drag them into a group, they become the last element in that group.

Why should I use it?

Reordered program groups ensure that datum references to features and constructed features can exist without program errors. You no longer need to manually cut and paste datums to a higher group in the **Inspection Navigator**.

Where do I find it?

Application	Inspection Programming
Prerequisite	An inspection file with an inspection feature and an inspection path on the inspection feature.
Resource bar	**Inspection Navigator**
Toolbar	**Navigator® Program Order View**
Menu	**Tools® Inspection Navigator® View® Program Order View**

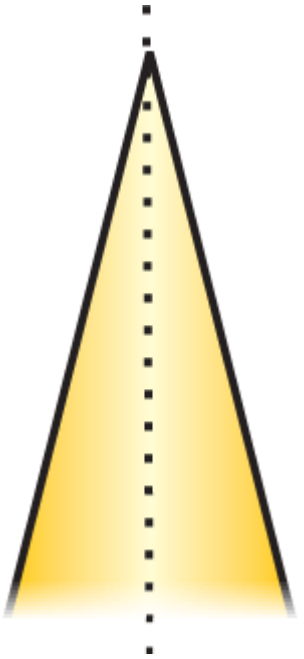
Inspection feature enhancements

What is it?

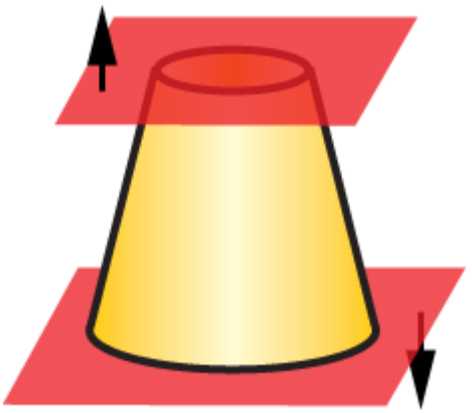
The **Plane Inspection Feature** and **Cone Inspection Feature** dialog boxes now include an **Extent** subgroup that lets you specify whether a plane or cone inspection feature created with face geometry is **Bounded** or **Unbounded**.

Although bounded and unbounded features do not appear different from other plane and cone features in the NX graphics window, bounded features differ from unbounded features in that they have finite planar extent boundaries. These generate BOUND events in your program output.

Any older inspection files containing planes or cones that were defined using face geometry will be automatically set to bounded when you generate and postprocess them.



Unbounded cone



Bounded cone

```
BOUND/F(TruncatedCone),F(Top),F(Bottom)
```

Why should I use it?

While plane and cone inspection features are unbounded by definition, tolerances such as position and total runout require bounded features.

Where do I find it?

Application	Inspection Programming
Prerequisite	You must create an inspection file.
Toolbar	**Feature® Plane**  **Feature® Cone** 
Menu	**Insert® Feature® Plane** or **Cone**
Inspection Navigator	Right-click® **Insert® Feature® Plane** or **Cone**

Constructed feature enhancements

What is it?

Constructed features are mathematically constructed from the characteristics of standard inspection features. The **Constructed Features** dialog box has been reorganized and enhanced to let you:

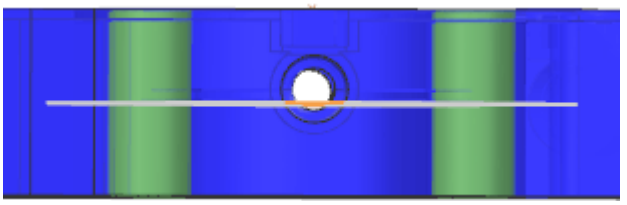
- Select a nominal feature, now termed **Design Feature**, prior to selecting the nominal and actual inspection features that define it.
- Create a new design feature for most types of constructed plane features without selecting a nominal inspection feature.
- Filter standard inspection features to be selected by feature type, and specify whether they should use actual or nominal values.

You can also select from a wide range of new and modified methods, now termed *construction forms*.

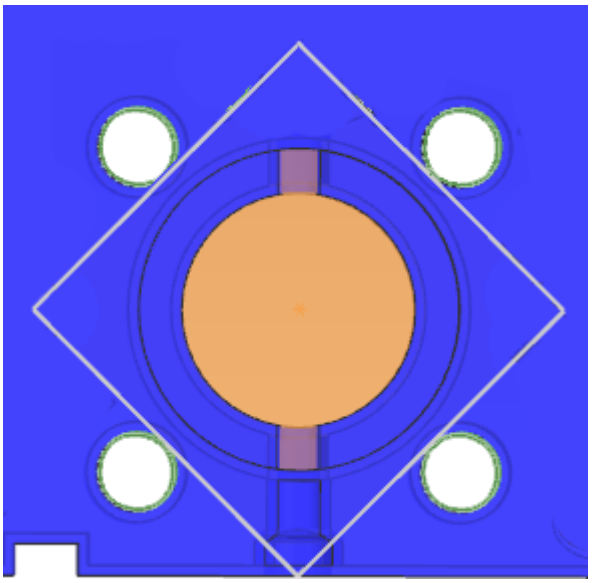
Construction form, design feature type	Description
Best Fit Point, line, plane, arc, circle, cylinder, cone, or sphere	Determines the best fit through or within any number of previously defined, point-reducible inspection features. If multiple solutions exist, the feature is constructed as close to the design feature as possible.

Example

The following shows a constructed plane feature in which the design feature does not use an existing inspection feature for its nominal. Using the actuals of four cylinder features, the calculated plane rests on the axial center points of the cylinders. Although it appears to be bounded by the cylinders in the graphics window, the constructed feature is actually unbounded.



Front



Top

Transform

Point, line, plane, arc, circle, cylinder, cone, or sphere

Transforms the constructed feature using the geometric attributes of a measured base feature and the following:

- The current Part Coordinate System (PCS)
- A nominal or actual PCS created using the **Alignment** command

Minimum, **Maximum**

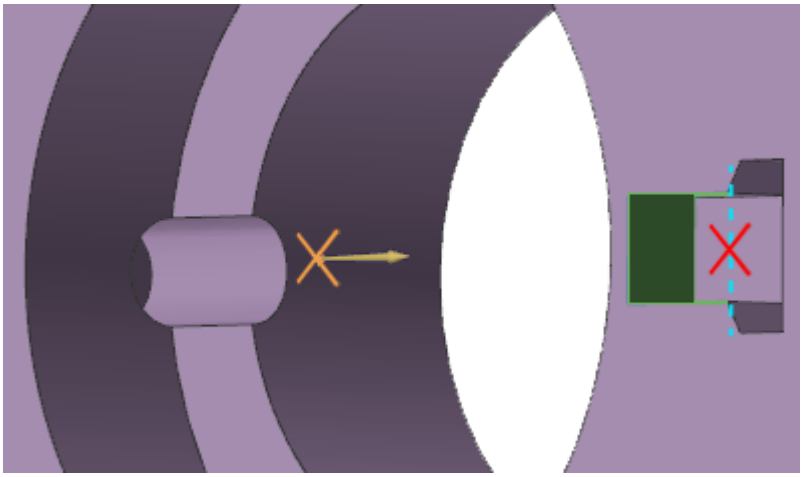
Point

Using a selected **Sub-Feature1** as an actual reference, creates a constructed point feature at the minimum or maximum distance to the subfeature in one of the following ways:

- Along a specified vector
- Radially from the center of an arc, circle, cylinder, cone, or sphere feature
- Using the ijk of another specified feature

Example

The following shows a constructed point feature that uses a nominal point at the center of a large circle as the design feature. The constructed feature uses the vector direction of the green plane feature, as well as the maximum distance of a closed slot feature. Actuals are derived from both the plane and the slot.



Move By Feature, **Move By Vector**

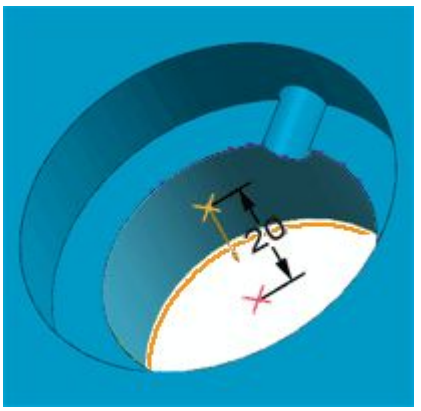
Point

Move by Feature creates a constructed point feature a specified distance from a nominal design feature, using the vector direction of another selected feature. Actuals are derived from the direction feature, a different measured point-reducible feature, or both.

Move by Feature works similarly but uses a specified, nominal direction vector rather than a direction feature.

Example

The following shows a constructed point feature moved by vector. A nominal point design feature, in orange, is projected 20 millimeters along a specified direction vector, using a point-reducible measured circle feature as the actual.



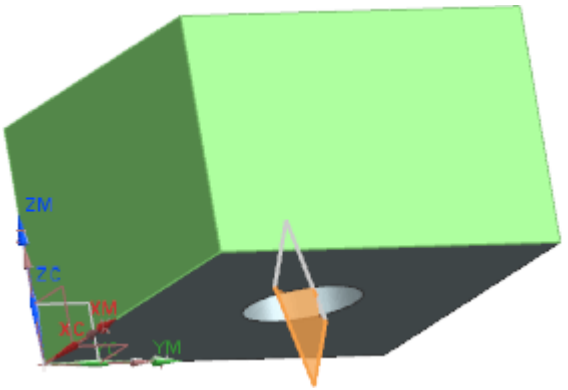
Middle

Point, line, or plane

Creates a constructed point, line, or plane feature at the midline, midplane, or midpoint between two inspection features.

Example

Below is a constructed plane feature in which the design feature does not use an existing inspection feature for its nominal. The plane design feature, in orange, lies midway between two plane features, both of which are actuals. Although it appears to be bounded, the constructed feature is actually infinite.

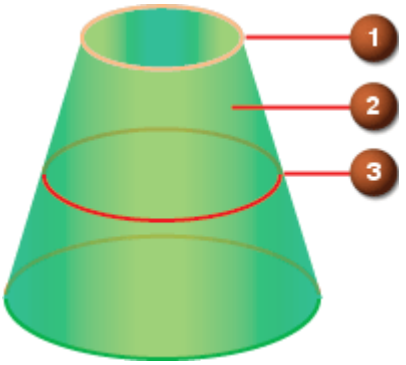


Cone Diameter

Circle

Creates a constructed circle feature at the location of the actual cone where the cone has the specified **Diameter** value.

Example



1 The design circle feature, with a nominal diameter of 8.0 millimeters.

2 The actual cone feature. A circle feature at the base has a diameter of 15.5.

3 The constructed circle feature. Since a **Diameter** of 11.75 is specified,

the constructed circle is midway along the cone:

```
(15.5 - 8) / 2 = 3.75
```

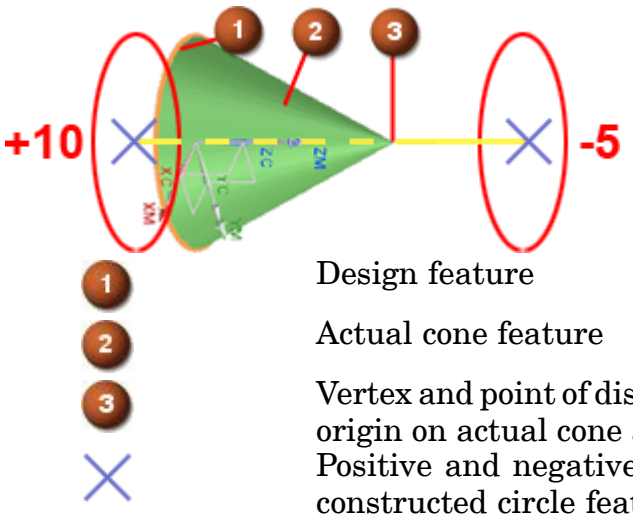
and

```
3.75 + 8 = 11.75)
```

Cone Distance Creates a constructed circle feature at a specified distance from the actual cone vertex along the actual cone axis toward the base of the cone. A negative distance creates the constructed circle in the opposite direction of the actual cone axis.

Circle

Example



Design feature

Actual cone feature

Vertex and point of distance origin on actual cone axis

Positive and negative constructed circle feature locations, based on **Distance**

Why should I use it?

Constructed features are often used to measure the size, location, and orientation of a theoretical feature or group of features relative to another feature or group of features. Use constructed features to derive a single feature from the point, line, and plane characteristics of other features.

Where do I find it?

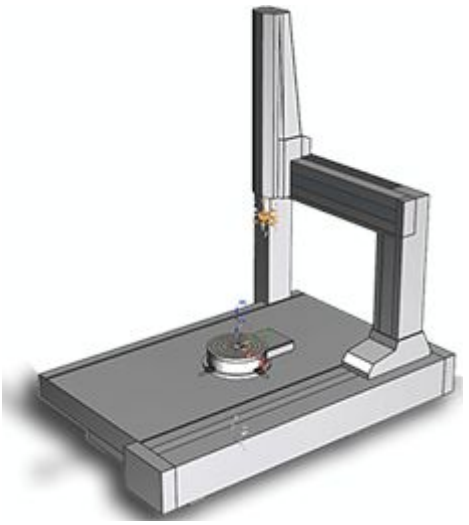
Application	Inspection Programming
Prerequisite	You must create an inspection file as well as any inspection features you need to create the constructed feature.
Toolbar	**Insert® Constructed Feature**

Menu	**Insert® Constructed Feature**
Inspection Navigator	Right-click® **Insert® Constructed Feature**

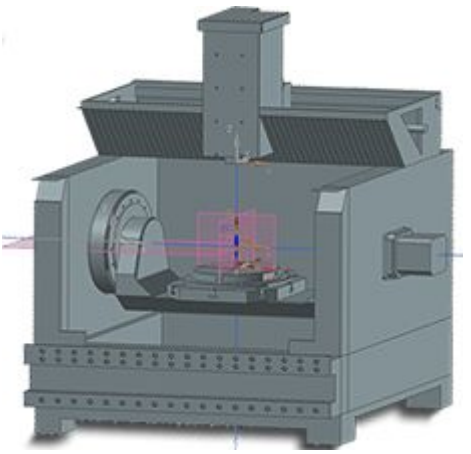
New virtual probe and machine models

What is it?

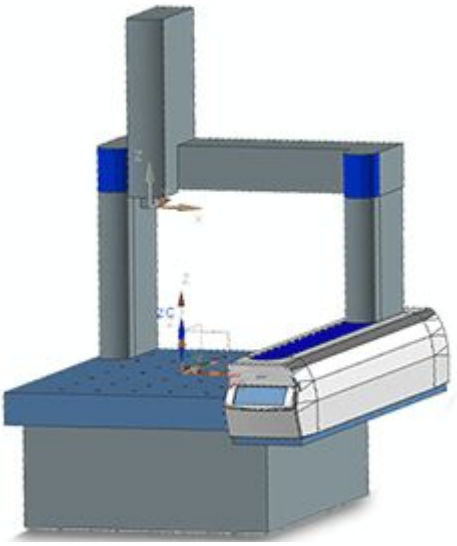
The Inspection Programming library features several new machine models, as well as a new probe and head.



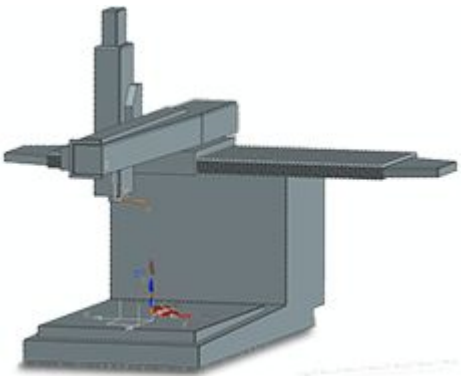
Brown and Sharpe X-cel 122010 with rotary table



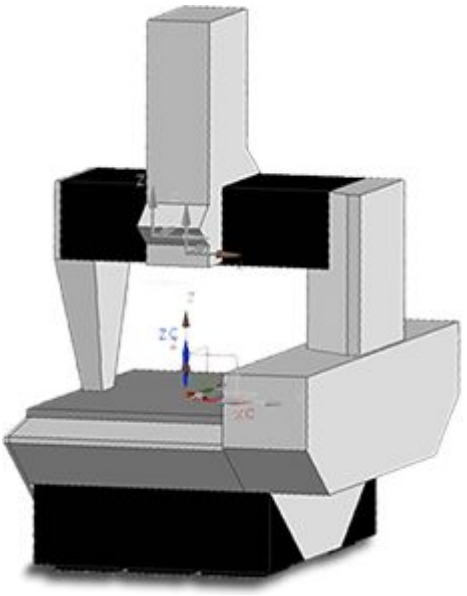
Zeiss Duramax



Hermle dual-axis rotary machine



Zeiss Prismo



Zeiss Contura

Star probe mounted to VAST fixed head

Why should I use it?

New machine models offer more options when you run inspection simulations. For detailed information on how to load machine components, see *Load a virtual CMM machine* in the NX 8.0 online help.

Where do I find it?

Application	Inspection Programming
Prerequisite	You must create an inspection file with no virtual machine loaded.
Inspection Navigator	In **Machine View**, double-click **GENERIC_MACHINE** and load the components.

Inspection path enhancements

Sub-Operations list enhancements

What is it?

The Sub-Operations list in the **Inspection Path** dialog includes three new columns:

- The probe **Tip** index—0 for single-tipped probes, 0 or higher for multi-tip probes.

- If applicable, the probe's **A Angle**, in degrees. The A angle usually rotates the probe barrel around the X-axis.

- If applicable, the probe's **B Angle**, in degrees. The B angle usually rotates the head around the Z-axis.

Why should I use it?

The probe tip and angles reported for each point or sub-operation let you quickly view the sensor strategies used for each point and sub-operation in the inspection path. You can select multiple points and sub-operations in the Sub-Operations list and then apply tip, probe, and angle settings globally to all those selected points and sub-operations.

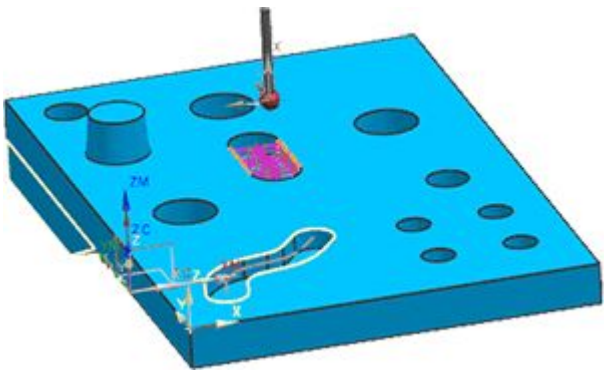
Where do I find it?

Application	Inspection Programming
Prerequisite	You must create an inspection file, and any necessary inspection features in PMI or Inspection Programming.
Toolbar	**Insert**® **Inspection Path** 
Menu	**Insert**® **Inspection Path**
Inspection Navigator	Right-click® **Insert**® **Inspection Path**

Enhanced approach logic and warnings

What is it?

The default programming logic related to how probes approach and retract from small holes and slots is enhanced. When creating inspection paths and sub-operations on such features, alerts will report potential problems in your process.



Alerts

Approach distance is too large for this feature.
Value will not be used.
Retract distance is too large for this feature.
Value will not be used.

Why should I use it?

Path warnings help you correct problems when you create an inspection path, rather than encountering them later during simulations.

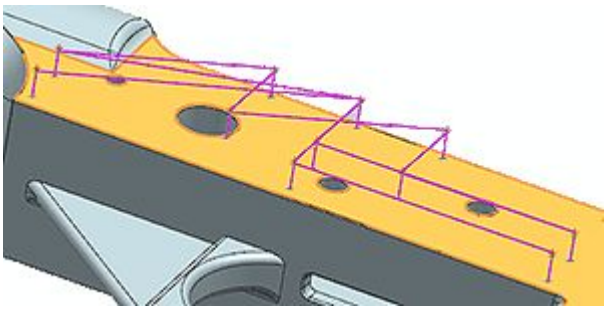
Where do I find it?

Application	Inspection Programming
Prerequisite	You must create an inspection file, necessary inspection features, and an inspection path on a small hole or slot.

Point set sub-operation enhancement

What is it?

A magenta inspection path preview now displays when you edit any parameter in a point set sub-operation except sensors. When you complete the sub-operation, the preview path is replaced by standard blue lines that indicate the completed point set path.



Where do I find it?

Application	Inspection Programming
Prerequisite	You must create an inspection file, necessary inspection features in PMI or Inspection Programming, and an inspection path.
Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation**  ® **Type** group® **Point Set** 

Scan sub-operation enhancements

What is it?

One parameter has been modified, and three new parameters have been added, to both scan curve and 5-Axis scan curve sub-operations.

Parameters	
Minimum Points	Formerly **Number of Points**, this parameter sets the minimum number of points to be scanned. This number may increase as the **Curvature Factor** increases.
 **Minimum Spacing**	Sets the minimum space that must exist between measurement points on the curve.
 **Maximum Spacing**	Sets the maximum space that must exist between measurement points on the curve.
 **Curvature Factor**	Sets the level of point density at areas that represent the greatest bends in the curve. A factor of 0 sets no increase in point density due to curvature. The greater the factor, the more point density increases at areas of greater curvature.

Why should I use it?

Increasing the curvature factor, as well as setting minimum and maximum spacing between points, lets you better retrieve measurement data from the most significant and variable areas of a curve. These settings retrieve less data from straighter, less variable areas, and produce more meaningful statistical results.

Where do I find it?

Application	Inspection Programming
Prerequisite	You must open a work part file with curved geometry, create an inspection file, and create a curve inspection feature.

Inspection Path dialog box	**Sub-Operations** group® **Add Sub-Operation**  ® **Type** group® **Scan Curve**  or **5 Axis Scan Curve** 

Chapter

8 PCB Exchange

Batch processing

What is it?

You can now process ECAD or NX files in a batch mode outside of NX. You need to create the file named *scenario* that contains the PCB Exchange commands you want to run. The *scenario* file must be located in the directory from where the *pcbsa.cmd* file is launched.

The *scenario* file must contain one command per line. You can use the following commands:

- `READ_ECAD` — Reads the ECAD model.
- `WRITE_ECAD` — Writes the loaded model to ECAD.
- `READ_NX` — Reads the NX model.
- `WRITE_NX` — Writes the loaded model to NX.
- `VALIDATE` — Validates the loaded model.

Example

The following is a sample *scenario* file to import an ECAD model into an NX model, and then validate it:

```
READ_ECAD
WRITE_NX
VALIDATE
```

To run a batch process, in a command shell, type:

```
pcbsa.cmd [filename]
```

`[filename]` is the model file name with no extension. File extensions for ECAD models and necessary PCB Exchange settings are defined in the PCB Exchange INI files. The NX model file extension should always be *.prt*.

Where do I find it?

The *pcbsa.cmd* file can be found in the *bin* folder in the following locations:

- *UGPCBXCHANGE* folder in the NX installation directory
- Location specified by MAYA_PCB_DIR

Chapter

9 Writing JT from NX

Export supplemental geometry PMI information to JT files

What is it?

You can now export PMI Region information, 3D Centerline, and Center Mark to JT files and view this information using Teamcenter Lifecycle Visualization.

Why should I use it?

To share the supplemental geometry and the attached PMI information in a multi-cad environment. You can also use this option to import the information back into NX.

Where do I find it?

Location in the configuration file	tessUG.config® ugConfig section® `activatesupplementalGeomPMI = true`