What's new in NX 9.0.2

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Contents

Chapter 1: Fundamentals

Licensing Options enhancements

What is it?

You can now use the **Licenses Attached to Bundle** list box to select frequently used NX application licenses that are available with the selected license bundle. You can free the selected license, if you do not want to use it. To free the license from the list box, select the license and click **Free Selected Licenses**.

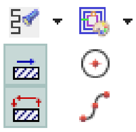
Where do I find it?

Application	Gateway
Command Finder	**Select License Bundles**

Chapter 2: Manufacturing

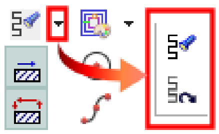
Displaying cutter paths

Display group



Use the two new options in the **Display** group of the **Home** tab to manage the display of tool paths when you select or edit an operation.

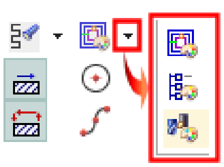
Display Drop-down



Use the **Display Tool Path**  option to show a tool path relatively quickly. The **Display** option supports the options in the display group.

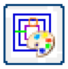
Use the **Replay Tool Path**  option to use all of the settings in the **Display Options** dialog box, including tool and path display options, replay speed, display of feed rates, arrows, and line numbers, and the generation options. You open the **Display Options** dialog box from the **Options** group when you are creating or editing an operation.

Coloring Drop-down



You can set the tool path display colors by motion type, by operation, or by tool.

The comparison images shown here are taken from one of the *sim06_mill_5ax_cam...* assemblies in your *...\MACH\samples\nc_simulation_samples* folder. The following three operations are selected in each image:

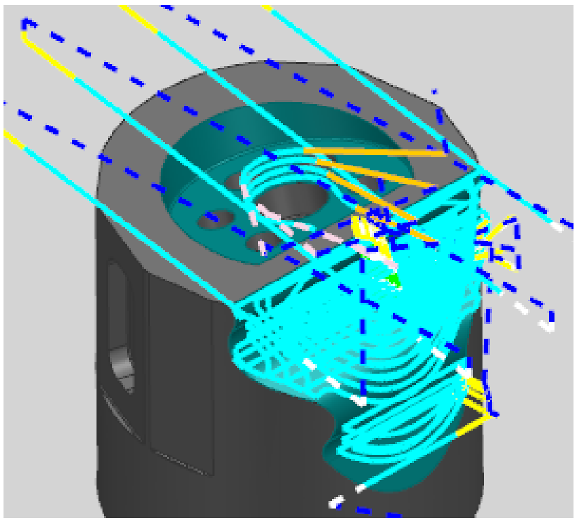


Motion Type

Use the **Motion type** option to retain the display behavior of versions previous to NX 9.0.2.

Tool path motions are displayed in the colors that you set in the **Options→Edit Display→Path Display Colors** dialog box. You can assign colors to different motions, such as engage, retract, rapid, and cutting moves.

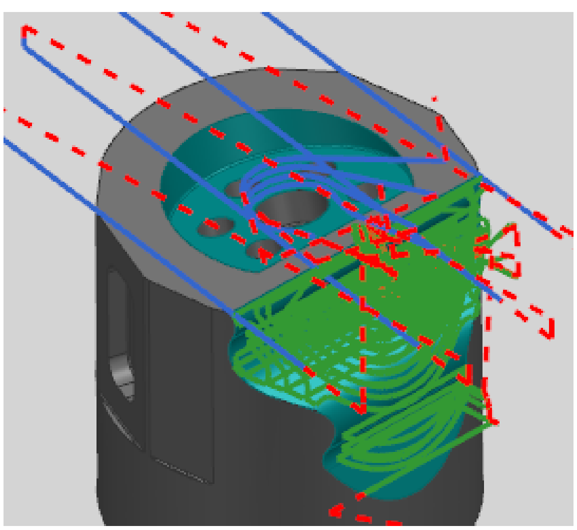
The example shows the default colors by motion type.



Tool

Use the **Tool** option to display the non-rapid moves of tool paths in a different color for each tool.

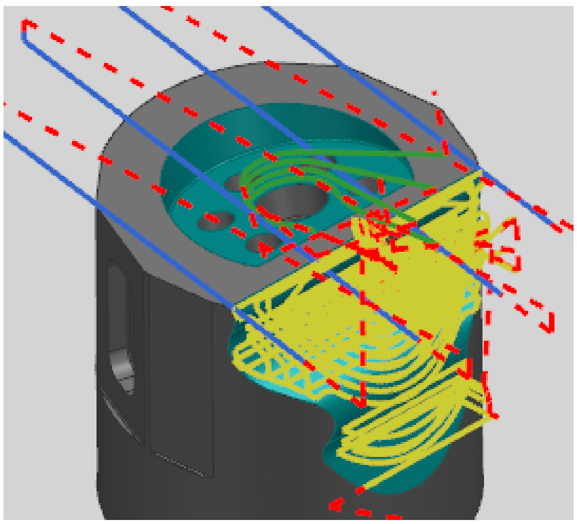
The example shows paths colored by tool. The first two paths are shown in a blue shade, and then there is a tool change. The third path, which uses a new tool, is shown in a green shade.



Operation

Use the **Operation** option to display the non-rapid moves of tool paths in a different color for each operation.

In the example, the path for the **FACE_TOP** operation is rendered in a blue shade, the path for the **CAVITY_TOP** operation is a green shade, and the path for the **CAVITY_MILL** operation is a yellow shade.



Customer defaults for tool and operation colors

The **Color by Tool or Operation** defaults control the default colors and order for tool path display by tool and by operation.

The **Rapid Motion**, **Color** and **Font** defaults control the font and color for rapid motions. Rapid motions are now red and dashed by default.

Tip

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

Where do I find it?

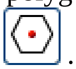
Application	Manufacturing
Prerequisite	Select operations in the **Operation Navigator** or edit an operation.
Ribbon bar	**Home** tab→**Display** group

Trimming tool paths in the Tool Path Editor

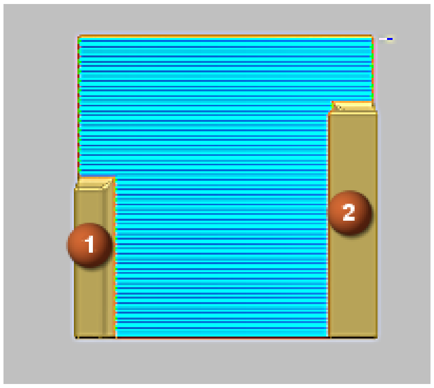
You can trim a tool path by selecting an area in the current graphics view plane. The selection tools include:

- Rectangle
- Lasso
- Polygon

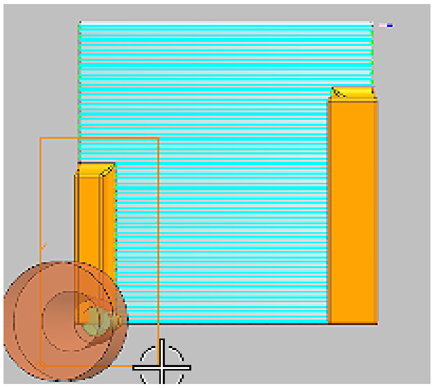
Tip

To add the polygon **Select by Polygon** method, use the **Command Finder** to find **Select by Polygon** . Right-click and choose **Add to Top Border Bar**. You can further customize the Top Border bar by dragging the icon into the **Selection** list.

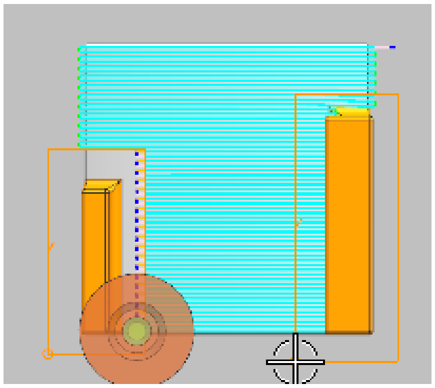
You can specify smooth connections between the truncated cutting motions. In the tool path shown, the walls marked (1) and (2) are colliding with the tool holder. You can trim those areas from the tool path.



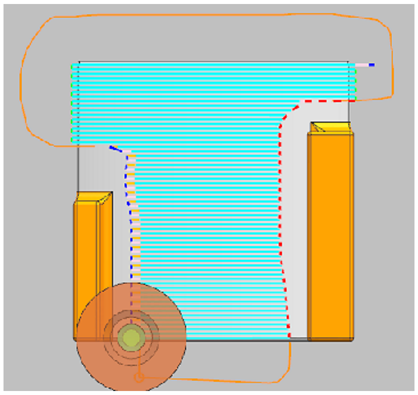
You can remove one area by trimming the motions that are inside a rectangle, as shown.



To complete the edit, repeat the trim using a second inside-rectangle trim.

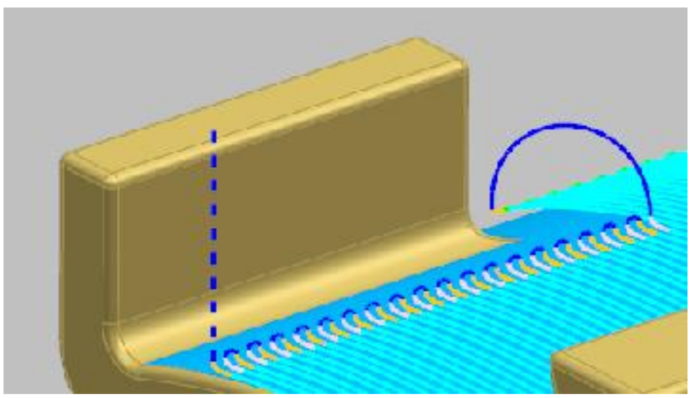


You can achieve the same result in a single trim operation by trimming outside a selected area using the **Lasso** (shown) or **Polygon** selection method.



Smooth connections between trimmed motions

The following figure shows the trimmed path, with the smooth connections option.



Where do I find it?

Application	Manufacturing
Prerequisite	You must edit a generated tool path.
Command Finder	Edit Tool Path
Operation Navigator	[Select operation]→right-click→**Tool Path**→**Edit**
Location in dialog box	**Edit Actions**→**Trim**  →**Trim Tool Paths** dialog box→**Trim Geometry** group→**Geometry** = **Selection in View** Choose the trim method on the Top Border bar.

Cut region control for Flowcut Reference Tool

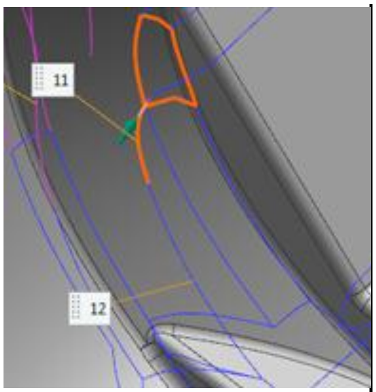
What is it?

You can subdivide the cut area for a **Flowcut Reference Tool** operation. To do this, set the **Cut Order** list to **User Defined** in the **Flow Cut Drive Method** dialog box. NX initially subdivides the cut area based on the steepness angle you provide.

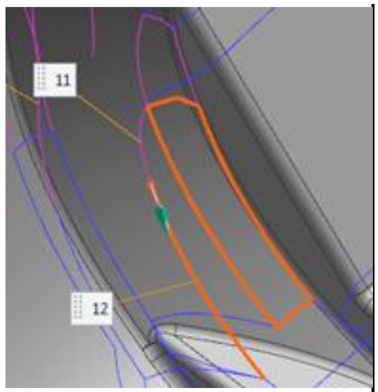
Region sizes

You control the region sizes. Merge or divide the regions as required. NX updates the cut region display as you make modifications.

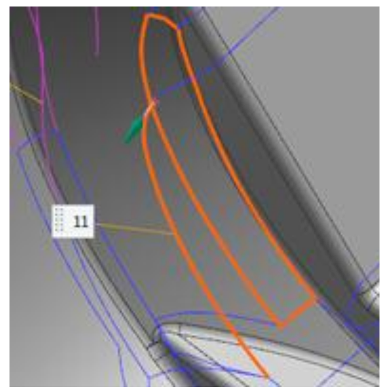
Example of merged regions



Region 11



Region 12



Regions merged as one

Region names when created:

<operation_name> _R_1

<operation_name> _R_2

When you divide regions, NX will append the region name with a sequential numerical value. For example, if you divide **Operation_R_1**, the resulting region names would be:

Operation_R_1_1

Operation_R_1_2

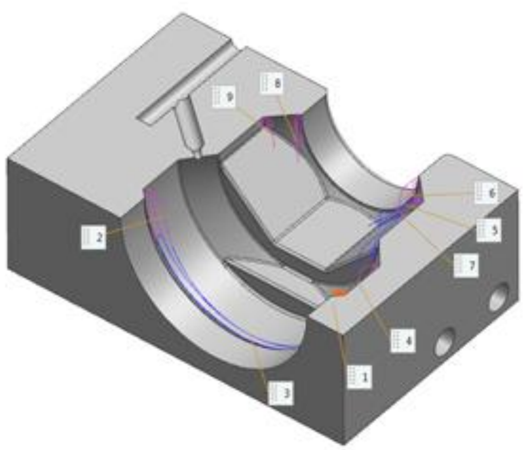
Cut regions and cutting sequence

You control the regions to cut and their cutting sequence. Delete regions to completely avoid cutting certain areas.

Cut Regions list example:

Name	Cut Order	Containment	Type
Operation Regions			
FLOWCUT_REF_TOOL_R_1	1		Flow
FLOWCUT_REF_TOOL_R_1	2		Flow
FLOWCUT_REF_TOOL_R_1	3		Flow
FLOWCUT_REF_TOOL_R_1	4		Flow
FLOWCUT_REF_TOOL_R_1	5		Flow
FLOWCUT_REF_TOOL_R_1	6		Flow
FLOWCUT_REF_TOOL_R_1	7		Flow
FLOWCUT_REF_TOOL_R_1	8		Flow
FLOWCUT_REF_TOOL_R_1	9		Flow

Cut order labels in the graphics display correlate to the **Cut Regions** list.



You can also do the following:

- Control the cut pattern and other tool path parameters within each cut region. Define a region as steep or non-steep, and NX applies the appropriate Flowcut Drive Method settings.
- Change the containment type for the region.
- Reverse the flowcut region cut direction. NX will not reverse an area region.

Editing updates

You can now do the following:

- Drag regions to change the cut order.
- Undo changes from the current editing session.
- Save the current cutting sequence. NX saves the edits with the operation each time it generates the operation, and uses the saved sequence to produce the updated tool paths.

Why should I use it?

Managing regions can improve cutting efficiency, improve the part surface finish, and improve cutter wear and performance.

Where do I find it?

Application	Manufacturing
Prerequisite	A **Flowcut Reference Tool** operation
Location in dialog box	**Flowcut Ref Tool** dialog box→**Drive Method** group→**Method** row→**Edit** **Flow Cut Drive Method** dialog box→**Output** group→**Cut Order**→**User defined**→**OK** **Flowcut Ref Tool** dialog box→**Actions** group→**Generate** →**Cut Regions** dialog box

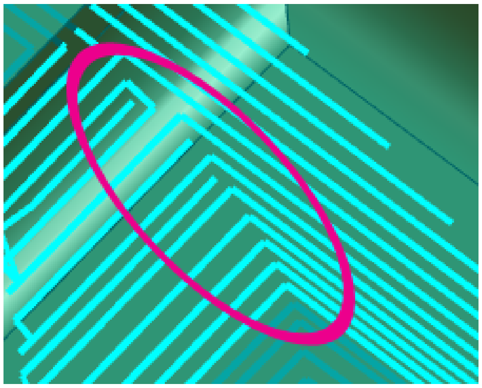
Corner smoothing

What is it?

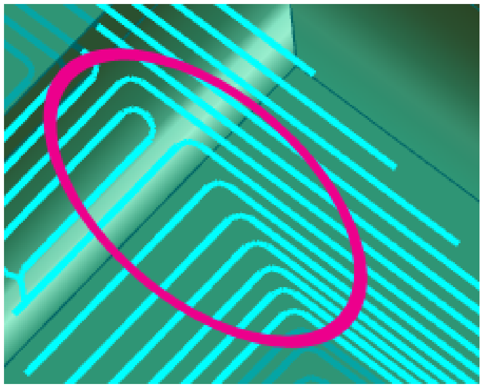
You can smooth corners in Zlevel cut patterns in **Flowcut Reference Tool** operations. To do this, you:

- Set steep and non steep regions to **Zlevel**.
- Select the **Corner Smoothing** cutting parameter.
- Specify a **Radius** size to be applied to the corners.

Previous versions of NX



NX 9.0.2



Why should I use it?

When you machine hard material, or machine at high speeds, consider adding fillets to all corners. Corner fillets can do the following:

- Prevent a sudden change in direction and cutter deflection which can cause excessive stress on the machine tool and cutter.
- Aid in tool path generation for Nurbs output because smooth transitions are easier to blend into Nurbs than sharp corners.

The **Corner Smoothing** option gives you better control of tool path output for finishing operations and improves machining efficiency.

Where do I find it?

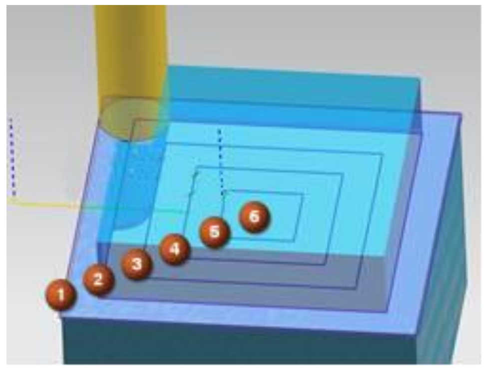
Application	Manufacturing
Location in dialog box	[**Flowcut Reference Tool** operation]→**Cutting Parameters**→**Corners** tab→**Path Shape in Corners** group→**Smoothing** row→**All Passes**

Automatic follow periphery pattern direction in cavity milling

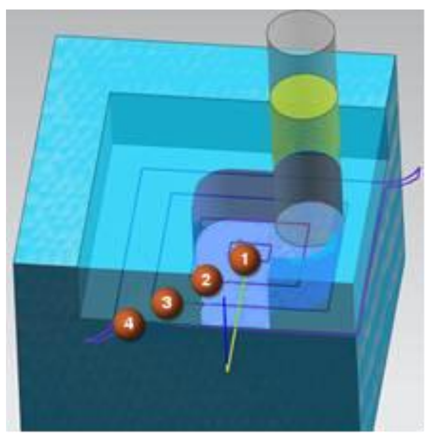
What is it?

You can choose the **Automatic** pattern direction option in a **Follow Periphery** cut pattern. The pattern direction will alternate between **Inward** and **Outward** per cut level and cut region condition. NX does the following.

- With an open periphery, the cutter approaches from outside the part and engages the material while working towards the center of the part.

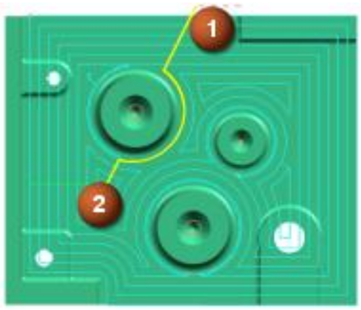


- With a partial or completely closed periphery, the cutter approaches from outside the part and the first initial entry machines a slot to reach the region start point. From there the toolpath continues to work its way outward.

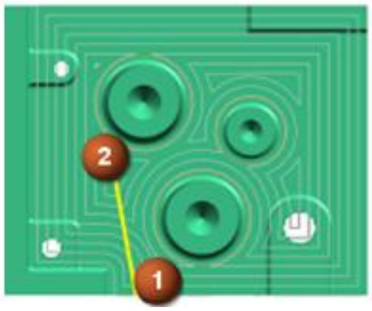


- For an **Outward** pattern with standing bosses or features, the entry point and path to the region start point are optimized.

Previous NX results



NX 9.0.2 results

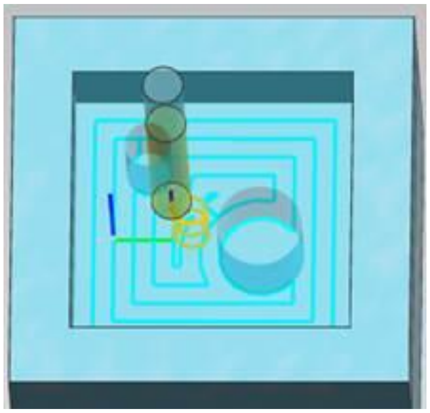


1 = Entry point

2 = Region start point

- Pockets, voids, or existing holes are considered for the open engage portion of the tool path.

Previous NX results



NX 9.0.2 results

Why should I use it?

By using the **Automatic** pattern direction option, you allow NX to optimize the created cavity milling tool paths for maximum efficiency, tool life and performance.

Where do I find it?

Application	Manufacturing
Location in dialog box	**Cutting Parameters** dialog box→**Strategy** tab→**Cutting** group→**Pattern Direction**→**Automatic**

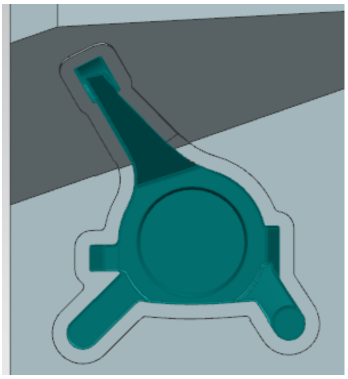
Using a boundary with the Area Milling drive method

Specify Trim Boundaries

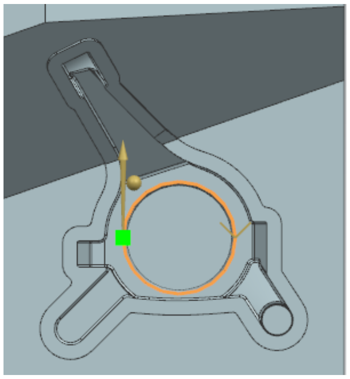
Use the **Specify Trim Boundaries** command to control cut areas in contouring operations that use the **Area Milling** drive method.

Contain the cut area by one or more loops of bounding objects. Boundaries can be edges, any type of curves including sketch curves, or points. Boundaries are projected onto the part surfaces along the normal vector of the boundary plane.

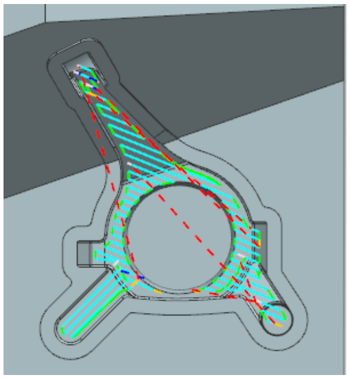
Edge as boundary



Cut Region

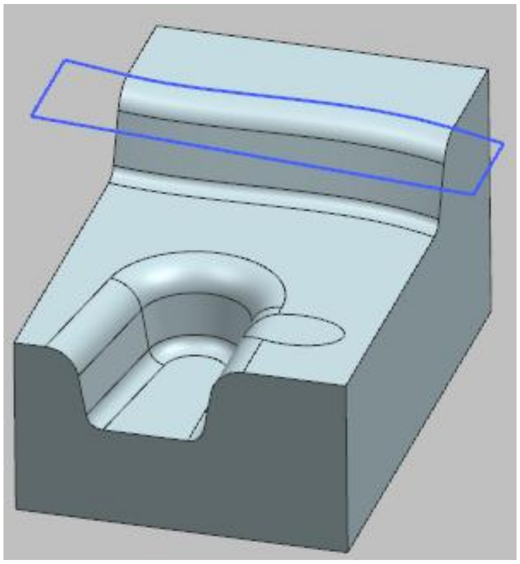


Boundary edge, trim inside

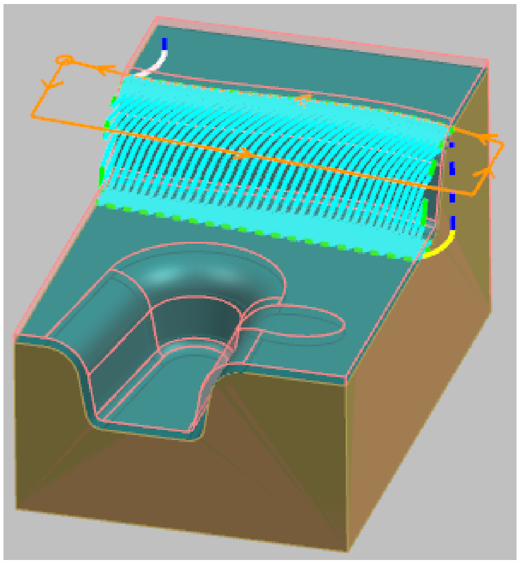


Tool Path

Sketch as boundary



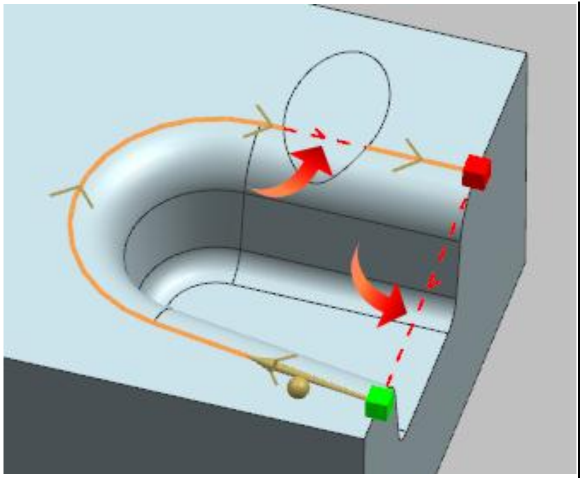
Boundary sketch, trim outside



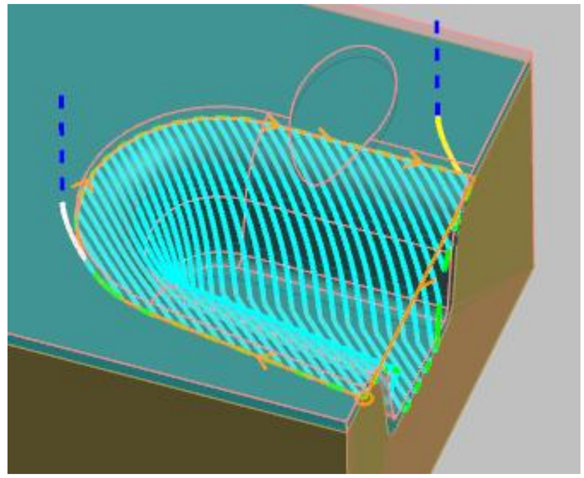
Tool Path

Closing gaps

You can leave gaps between curves or edges. The gaps are automatically closed by straight lines.



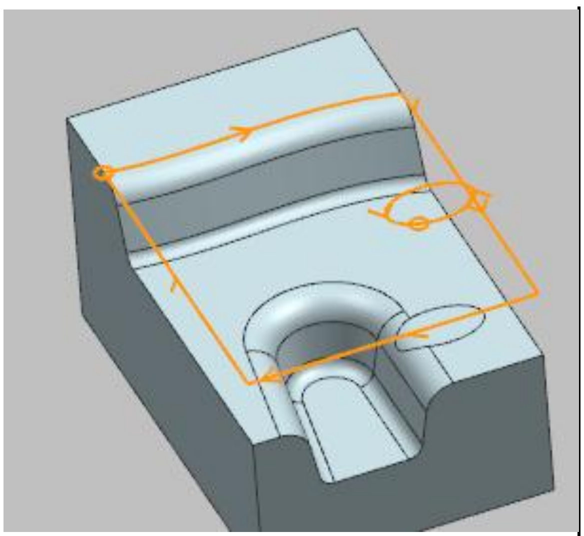
The arrows indicate closed gaps, boundary trim outside



Tool Path

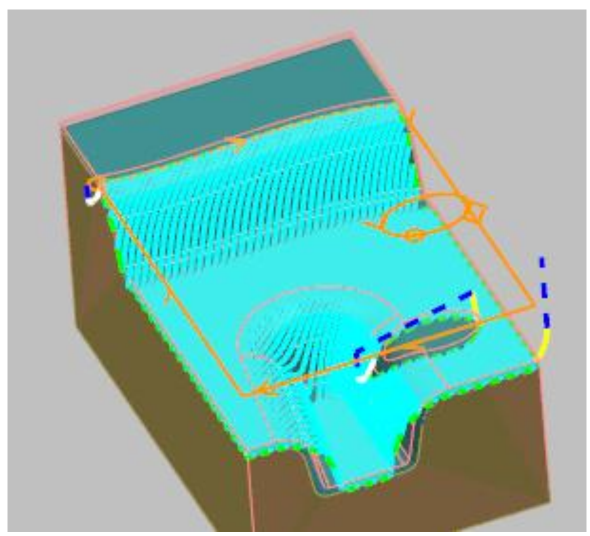
Nested boundaries, no overlap

You can define nested containment boundaries provided they indicate a clear trimming area and do not overlap.



Outer boundary trim outside

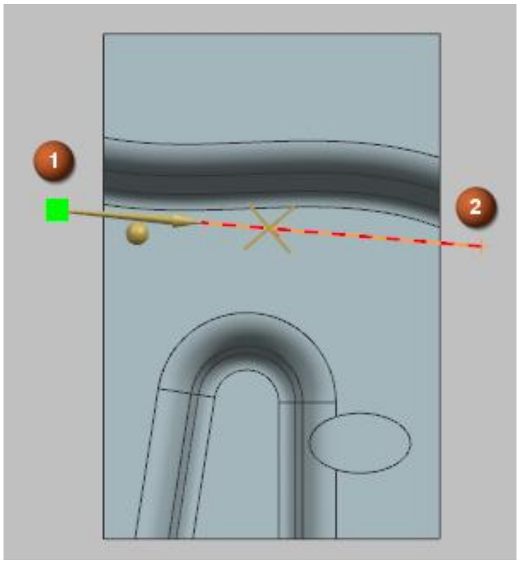
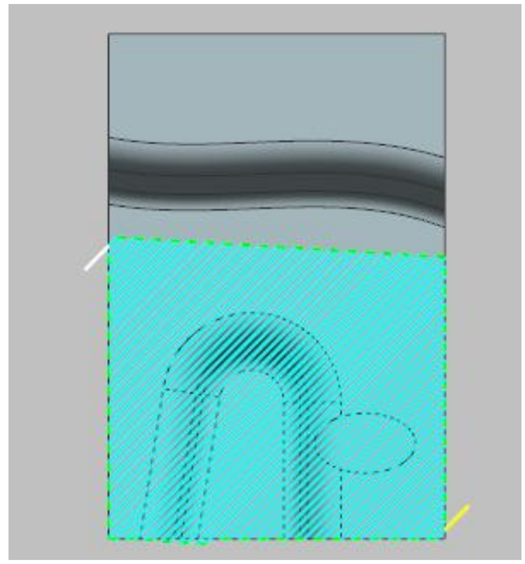
Inner boundary, trim inside



Tool Path

Linear open boundary

You can define open boundaries provided that the open ends project off the part.

	
Mouse clicks at (1) and (2) define two points, off part, as an open boundary	Tool Path

Trim boundary options

The trim boundary options are always available in the **Contour Area** operation subtype. They appear in the following operation subtypes after you select the **Area Milling** drive method :

- **Fixed Contour**
- **Contour Surface Area**

Contact tool position

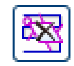
When you specify the **Contact** tool position, NX moves the tool to contact the projected boundary in such a way that the entire area is cut.

Area milling geometry with boundaries

In the **Geometry** view of the **Operation Navigator**, you can create a **MILL_AREA** geometry group and include trim boundaries in the definition. You can then reuse the geometry area in multiple operations.

NX displays this type of geometry group in the initial **Geometry** list only when you choose the **Contour Area** operation subtype. If you create **Fixed Contour** or **Contour Surface Area** operation subtypes, you must edit the drive method to **Area Milling** before the area mill geometry appears in the **Geometry** list.

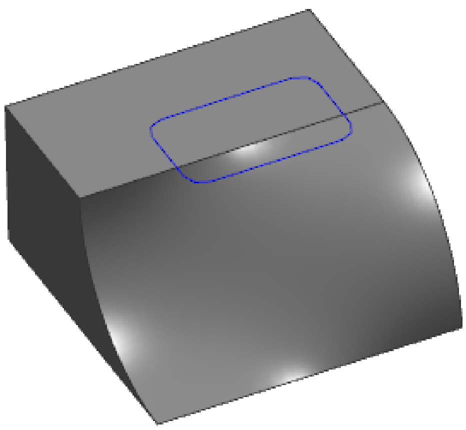
Where do I find it?

Application	Manufacturing
Prerequisite	You must have a contouring operation that uses the **Area Milling** drive method.
Location in dialog box	**Geometry** group→**Specify Trim Boundaries** 

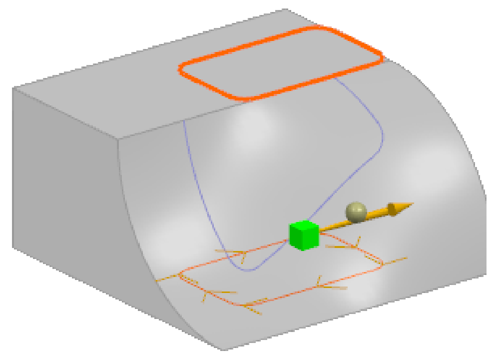
Contact tool position on Area Milling boundaries

When you specify the **Contact** tool position, NX moves the tool to contact the projected boundary in such a way that the entire area is cut.

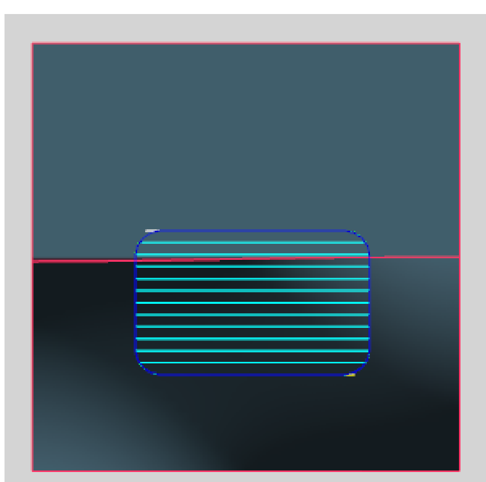
The part for this example has a surface that has both concave and convex regions. A sketched rounded rectangle is the boundary.



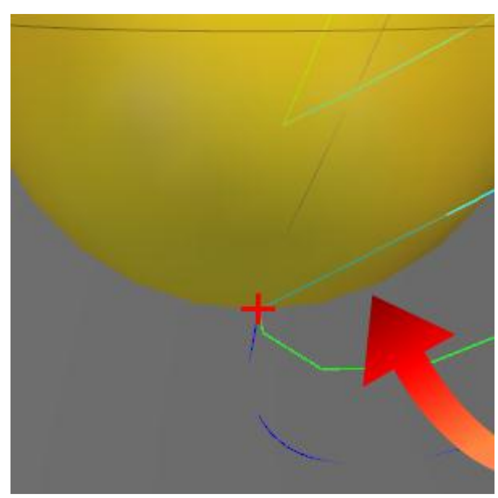
When projected along the tool axis, the Z-axis in this example, the boundary encompasses both concave and convex surface areas.



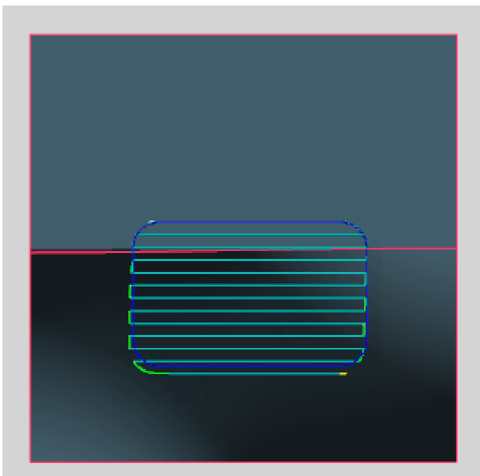
As seen along the tool axis, a tool path with the **On** tool position moves the tool tip strictly within the boundary.



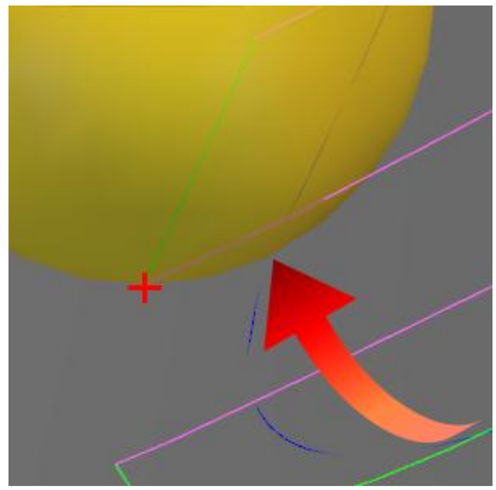
When you use the **On** tool position, in steep areas, small concave areas are not completely cut, and small convex areas are cut outside of the boundary. This happens because when the tool tip is positioned over the boundary, the point of contact of the tool with the face is some distance from the boundary.



As seen along the tool axis, a path with the **Contact** tool position moves the tool tip beyond the boundary, or stops short of the boundary, depending on the nature of the surface slope.



The **Contact** option positions the tool tip such that the point of contact of the tool with the face lies on the boundary. This ensures that the tool contacts all points that lie inside the boundary, and does not contact points that lie outside of the boundary.

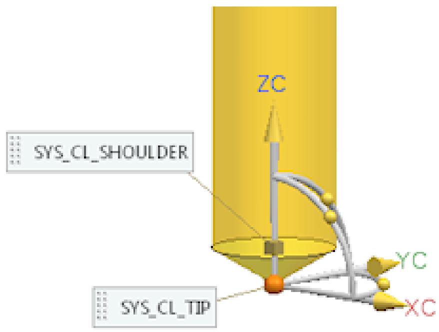


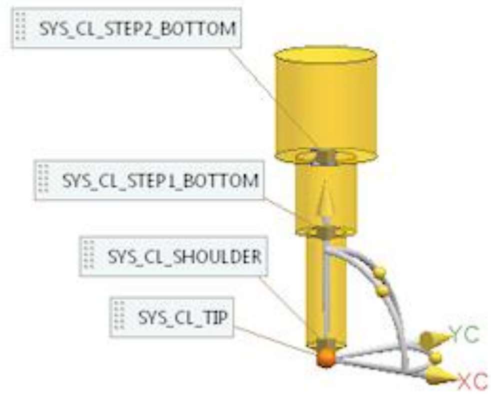
Where do I find it?

Application	Manufacturing
Prerequisite	Create or edit an operation that uses the **Area Milling** drive method, and create or edit boundaries to contain the tool motion.
Operation dialog box	**Specify Trim Boundaries** 
Location in dialog box	**Trim Boundaries** dialog box→**Boundaries** group→**Tool Position** list→**Contact**

Tracking points for drilling tools

A tracking point is the internal reference point on the tool that NX uses to calculate tool paths.

<table>
<tr>
<td>Drilling tools have two system-defined tracking points at the center line SYS_CL_SHOULDER and SYS_CL_Tip.


</td>
<td>Step drills have an additional system-defined tracking point at the center line of each step.


For example, SYS_CL_STEP1_BOTTOM is located at the bottom of the first step.</td>
</tr>
</table>

You can also define additional tracking points anywhere on the tool to use in your operation.

Using the tracking points in the drilling operation

You can specify separate tracking points for the non cutting and the cutting moves of a drilling operation.

- To ensure that the tool tip clears the part, select the tracking point at the tool tip for non cutting moves.
- To ensure that a position on the tool reaches a certain depth, select the tracking point that corresponds to that position. For example, select the shoulder of a drill, or a specific shoulder of a step drill.

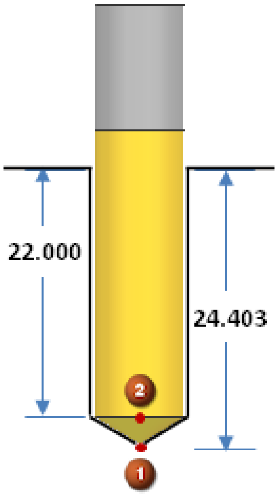
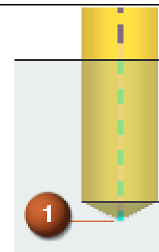
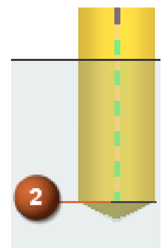
NX automatically changes to the correct tracking point for each type of move. For example, the drilling operation could use the following system-defined tracking points:

- **SYS_CL_Tip** at the tool tip for all positioning and traversal moves.
- **SYS_CL_SHOULDER** at the tool shoulder for the cutting moves within the drilling cycle.

If the cycle output contains rapid moves, such as those for Break Chip or for interrupted holes, the drilling operation would then use:

- **SYS_CL_Tip** for traversal, approach, and departure moves.
- **SYS_CL_SHOULDER** for engage, cutting, and retract moves within the drilling cycle.

Example outputs for different drilling cycle tracking points

		
		1 **SYS_CL_TIP** output:
		`GOTO/-25.352,89.572,-24.403`
		
		2 **SYS_CL_SHOULDER** output:
		`GOTO/-25.352,89.572,-22.000`

Where do I find it?

Application	Manufacturing
Prerequisite	Drilling tool
Location in dialog box	Define the tracking point: Drilling tool dialog box→**More** tab→**Tracking** group→**Tracking Points**→**Tracking Points** dialog box Set the tracking point for non cutting positioning moves: [Drilling operation dialog box]→**Tool** group→**Cutter Compensation** subgroup→**Tracking Data** list **Note** To select a specific tracking point, you must customize the drilling operation dialog box to include the cutter compensation **Tracking Data** list. By default, NX uses the **None** option, which outputs the tool path at the tool tip centerline.

	Set the tracking point for the drilling cycle: [Drilling operation dialog box]→**Path Settings** group→**Cycle Tracking Data** list

Machining holes with fewer operations

You can group through holes and blind holes to machine them together. You save on programming time because you do not have to program the through holes and blind holes separately. Use the **Predefined** option of the **Group Features** command to group features that have identical diameters, thread pitch, and number of steps. When it groups the features, NX ignores the values of all depth attributes, including tolerances, chamfers and thread lengths.

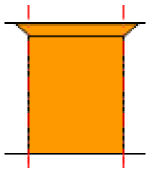
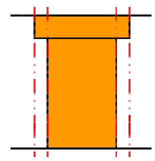
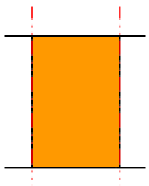
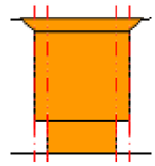
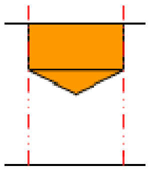
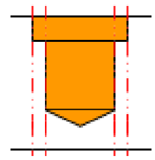
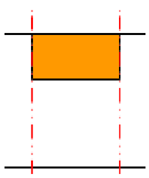
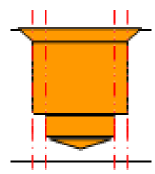
How it works

In NX, through holes are **STEPxHOLE** features and blind holes are **STEPxPOCKET** features, where x indicates the number of steps in the feature. When you group features using the **Predefined** option, NX:

- Includes all **STEP1HOLE** and **STEP1POCKET** features with identical diameters in an **FG_STEP1HOLE_STEP1POCKET** feature group.
- Includes all **STEP1HOLE_THREAD** and **STEP1POCKET_THREAD** features with identical diameters and thread pitch in an **FG_STEP1HOLE_THREAD_STEP1POCKET_THREAD** feature group.
- Repeats the process for features with multiple steps.

If additional feature groups are required for holes with different diameters, NX creates the feature groups and names them **FG_STEP1HOLE_STEP1POCKET_1**, **FG_STEP1HOLE_STEP1POCKET_2** and so on.

The following example shows different hole feature profiles, and how NX groups the hole features together.

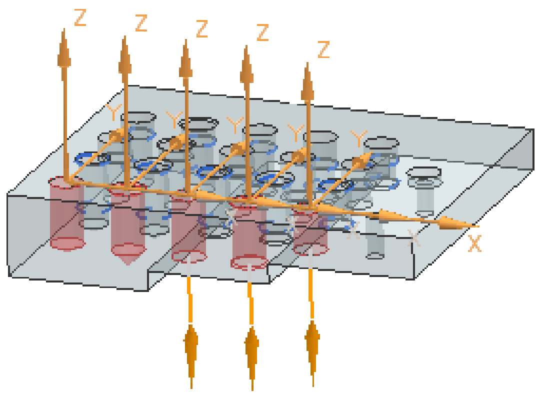
Features to group			
	STEP1HOLE_1		STEP2HOLE_5
	STEP1HOLE_2		STEP2HOLE_6
	STEP1POCKET_3		STEP2POCKET_7
	STEP1POCKET_4		STEP2POCKET_8

Features in the Machining Feature Navigator after grouping

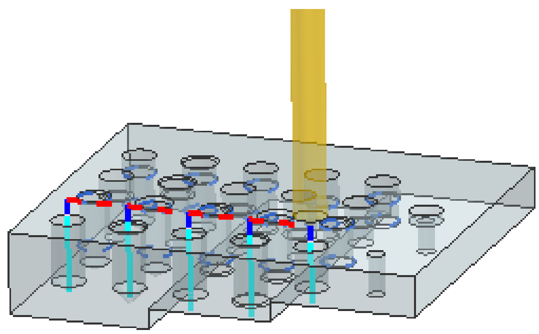
- WORKPIECE
  - FG_STEP1HOLE_STEP1POCKET
    - STEP1HOLE_1
    - STEP1HOLE_2
    - STEP1POCKET_3
    - STEP1POCKET_4
  - FG_STEP2HOLE_STEP2POCKET
    - STEP2HOLE_5
    - STEP2HOLE_6
    - STEP2POCKET_7
    - STEP2POCKET_8

Sample part

Features selected for **FG_STEP1HOLE_STEP1POCKET**



Drilling operation for **FG_STEP1HOLE_STEP1POCKET**



The **Group Features** command also has the following options:

- **None** groups features whether or not the attributes are identical.
- **All** groups features only if all of the attributes are identical.

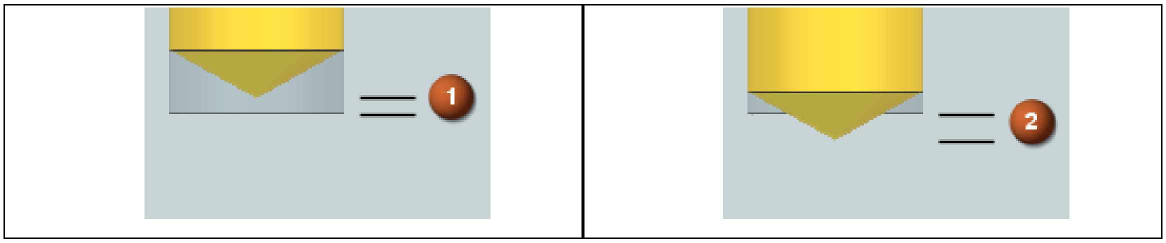
Where do I find it?

Application	Manufacturing
Machining Feature Navigator	Right-click in the background→**Group Features** Grouping selected features Right-click selected features→**Group Features**
Location in dialog box	**Group by** group→**Identical Attributes** list →**Predefined**

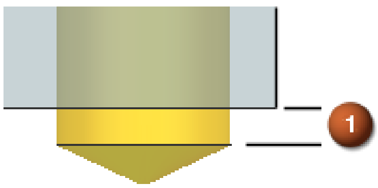
Enhancements to control drilling depth

You can control drilling depth for drills and step drills in the following ways.

- Apply a positive (1) or negative (2) bottom stock to blind holes.



- Apply a bottom offset (1) to through holes.



- Use the **MODEL_DEPTH** option to drill to the depth where the tool contacts the part.

  Use this method to control cutting depth for hole features where depth attributes are not relevant or not desired.

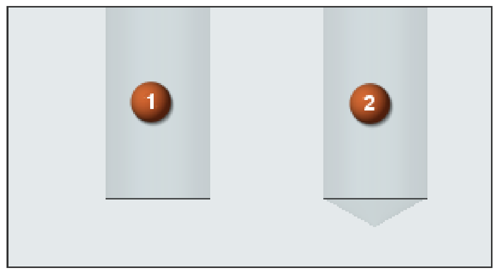
- Use the **Position Tool Shoulder** option to control whether the tool contact point is at the tool tip or the tool shoulder.

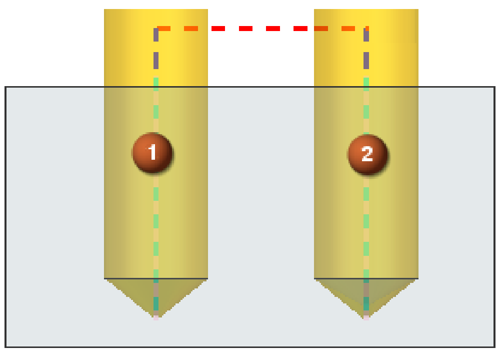
  Note

  The **Position Tool Shoulder** option is separate from the **MODEL_DEPTH** option.

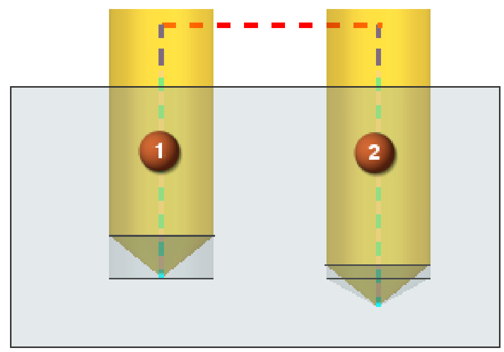
Drilling to tool contact depth

Use the **MODEL_DEPTH** option to drill to where the tool contacts the part. You control the drilling depth by choosing whether the tool contact point is at the tool tip or the tool shoulder. In the following example, (1) is a hole modeled with a flat bottom and (2) is a hole modeled with a tip angle.





Position Tool Shoulder = ☑

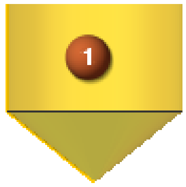
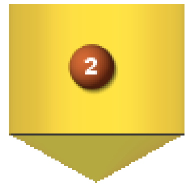
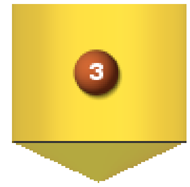
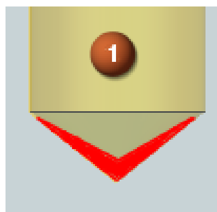
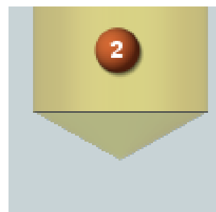
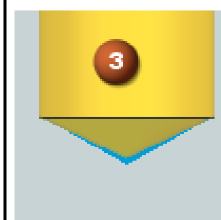
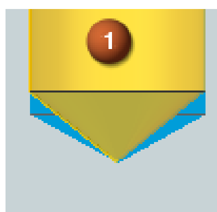
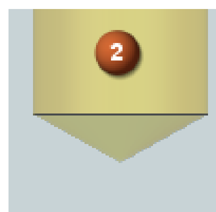
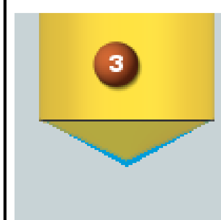


Position Tool Shoulder = ☐

If you drill the hole with a tool that does not match the tip angle of the modeled hole, the operation will either leave material or gouge the part.

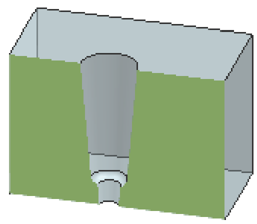
- If you use the tool shoulder to position the tool, and the drill tip angle is less than the modeled tip angle, the operation gouges the part.
- If you use the tool tip to position the tool, and the drill tip angle is greater than the modeled tip angle, the operation drills as deeply as it can without gouging the part.

The following example shows a hole feature modeled with a 122° tip.

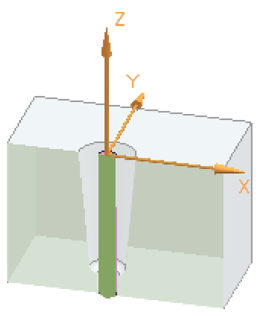
Hole	102° drill tip	122° drill tip	130° drill tip
			
Position to tool shoulder	 Gouges	 Exact match	 Leaves material
Position to tool tip	 Leaves material	 Exact match	 Leaves material

Conical holes

Use the **Model Depth** and **Bottom Stock** options to help you drill conical holes. The following example shows the in-process features for the operations used to rough a tapered hole.



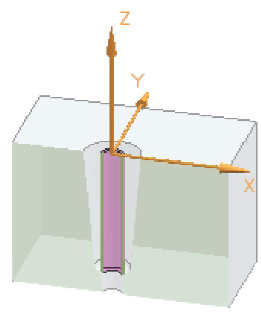
Volume to drill: 5 mm through hole with conical taper from 8 mm to 13 mm



4 mm drill

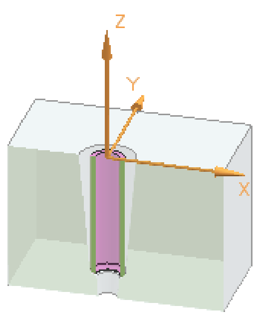
Bottom Stock = 0

Bottom Offset = 1.0



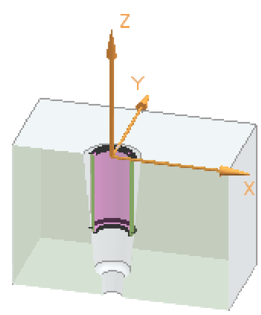
5.5 mm drill

Bottom Stock = 0.1



8 mm drill

Bottom Stock = 0.1



10 mm drill

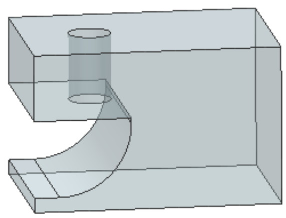
Bottom Stock = 1.0

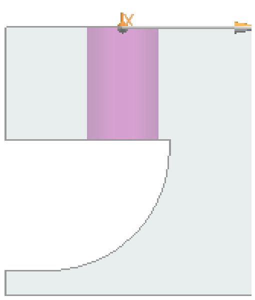
Protecting part geometry below a through hole

Use the **Model Depth** and **Bottom Stock** options also for through holes when you need to extend the depth of the in-process feature to the part geometry.

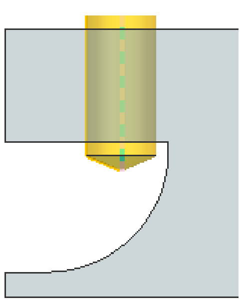
> **Note**
>
> When you select the **Model Depth** feature geometry setting, and the in-process feature of a through hole extends to the part geometry, NX ignores the **Bottom Offset** cutting parameter setting. NX uses the **Bottom Stock** setting instead.

The following example shows the in-process features and resulting tool path for different **Machining Area** and **Bottom Stock** option settings.

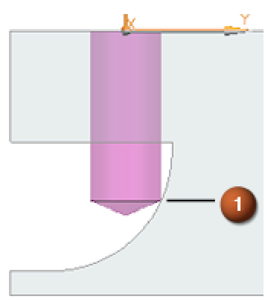




Machining Area = **FACES_CYLINDER_1**



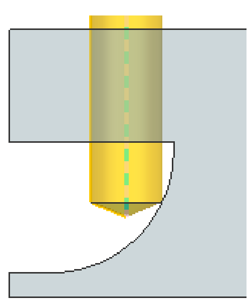
Bottom Offset = 2



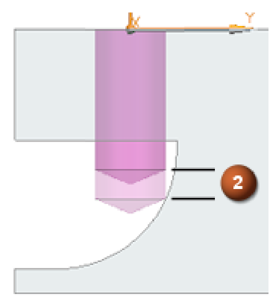
Machining Area = **MODEL_DEPTH**

1 **Bottom Stock** = 0

(contact point)

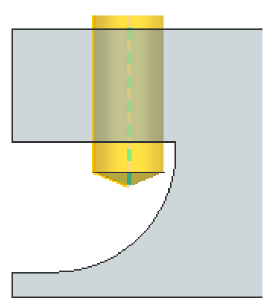


Bottom Offset = 2



Machining Area = **MODEL_DEPTH**

2 **Bottom Stock** = 5



Bottom Offset = 2

Where do I find it?

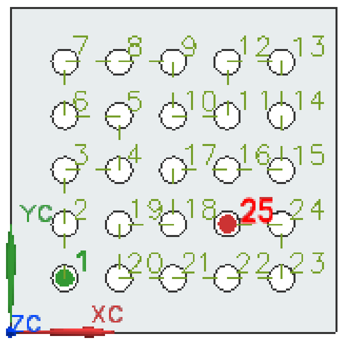
Application	Manufacturing
Location in dialog box	Drilling to model depth **Feature Geometry** dialog box→**Common Parameters** group→**Machining Area** list →**MODEL_DEPTH** Positioning to the tool shoulder or tool tip **Feature Geometry** dialog box→**Common Parameters** group→**Cutting Parameters** subgroup→**Position Tool Shoulder** Adding bottom stock to blind holes **Feature Geometry** dialog box→**Common Parameters** group→**Cutting Parameters** subgroup→**Bottom Stock** Adding bottom offset to through holes [Drilling operation] dialog box→**Cutting Parameters**→**Cutting Parameters** dialog box→**Strategy** tab→**Extend Path** group→**Bottom Offset**

Enhancements to optimize drill sequencing

You can optimize the drilling sequence in the following ways:

- Machine parts with holes arranged in a grid using a zig or zig-zag pattern to minimize tool travel. Whenever possible, NX keeps the transition motions parallel to the axis or vector that you specify as the primary pattern direction.
- Reverse the order of the listed hole or boss locations.
- Select a hole feature and make it the start or end location of the cutting sequence.

To see the machining order without generating the operation, preview the hole sequence.

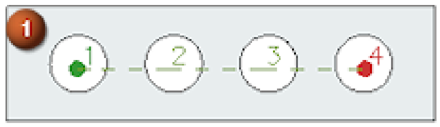


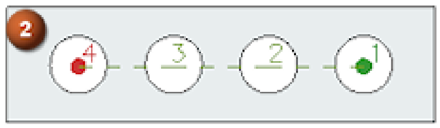
Defining the hole sequence

You can use the hole sequence defined in the **HOLE_BOSS_GEOM** geometry parent, or modify the sequence for an individual operation.

Operation Navigator — Geometry

- **WORKPIECE**
  - **HOLE_BOSS_GEOM**Defines the default hole sequence (1).
    - **SPOT_DRILLING** Uses the default sequence.
    - **DRILLING**Modified to use a reversed hole sequence (2).
    - **COUNTERSINKING**Uses the default sequence.





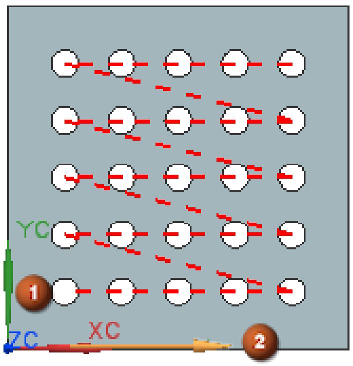
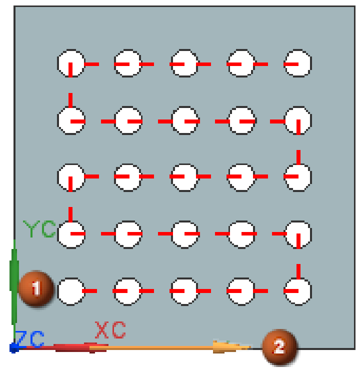
Use the **Reload List From Parent** sequencing option in the operation after changes in the parent sequence.

> **Note**
>
> The options in the **Sequence** group in the **Hole or Boss Geometry** and **Feature Geometry** dialog boxes always display the default settings. The sequencing options are *actions* that NX execute when you click **Reorder List**. NX saves the resulting sequence without saving the settings.

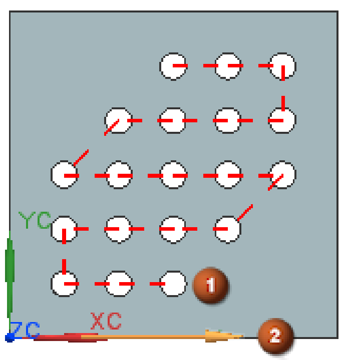
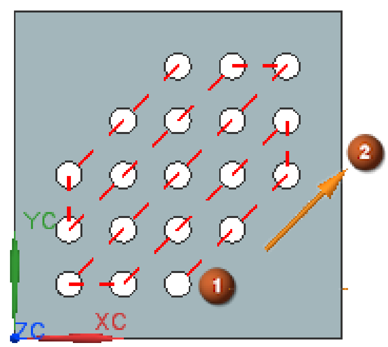
Machining holes using a zig or zig-zag pattern

To minimize tool travel, machine parts with holes arranged in a grid using a zig or zig-zag pattern. Whenever possible, NX keeps the transition motions parallel to the axis or vector that you specify as the primary pattern direction.

The following example compares a zig pattern and a zig-zag pattern with the same start (1) point. The primary pattern direction (2) is along the X-axis.

Zig pattern	Zig-zag pattern
	

The following example compares two zig-zag patterns with the same start (1) point, but different primary pattern directions (2).

Primary direction along X-axis	Primary direction along specified vector
	

Changing the start or end points

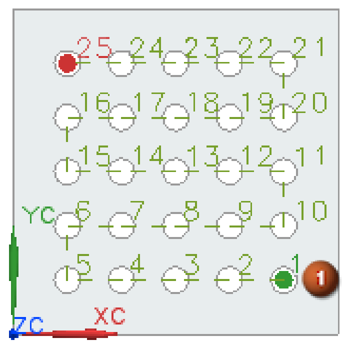
To make a feature the start or end location, select it and click **Move to top**  or **Move to Bottom** .

Caution

Specifying an end location when using a **Zig** or **Zig-zag** pattern is not recommended. NX determines the appropriate end location for you.

The following example compares the **Hole or Boss Geometry** dialog box list and preview for a hole sequence before and after changing the start location.

Hole sequence with **STEP1HOLE_1** as the start (1) location.

Item	Name	
1	STEP1HOLE_1	
2	STEP1HOLE_2	
3	STEP1HOLE_3	
4	STEP1HOLE_4	
5	STEP1HOLE_5	
.	.	
.	.	
.	.	
25	STEP1HOLE_25	

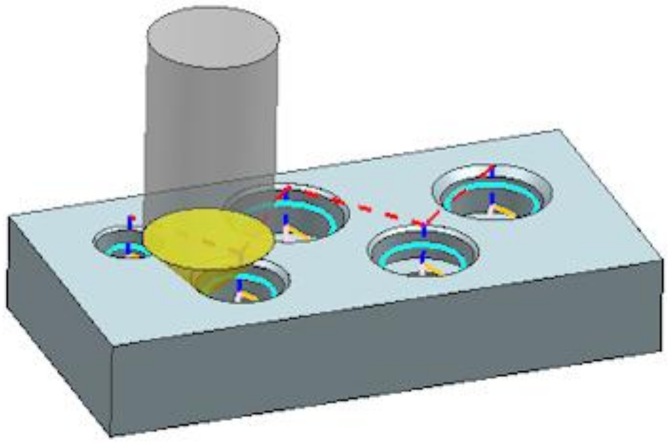
Hole sequence after moving **STEP1HOLE_5** to the start (1) location.

Item	Name	
1	STEP1HOLE_5	
2	STEP1HOLE_4	
3	STEP1HOLE_3	
4	STEP1HOLE_2	
5	STEP1HOLE_1	
.	.	
.	.	
.	.	
25	STEP1HOLE_21	

Where do I find it?

Application	Manufacturing
Location in dialog box	Defining the sequence for multiple operations **Hole or Boss Geometry** dialog box Defining the sequence for a single operation [Drilling operation] dialog box→**Geometry** group→**Specify Feature Geometry**→**Feature Geometry** dialog box Changing the sequence start or end location **Hole or Boss Geometry** or **Feature Geometry** dialog box→**Feature** group→**List** subgroup→**Move to top** or **Move to Bottom** Zig or zig-zag pattern sequence **Hole or Boss Geometry** or **Feature Geometry** dialog box→**Sequence** group→**Optimization**→**Primary Direction**

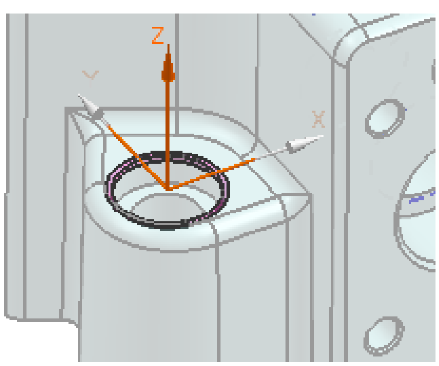
Milling hole chamfers



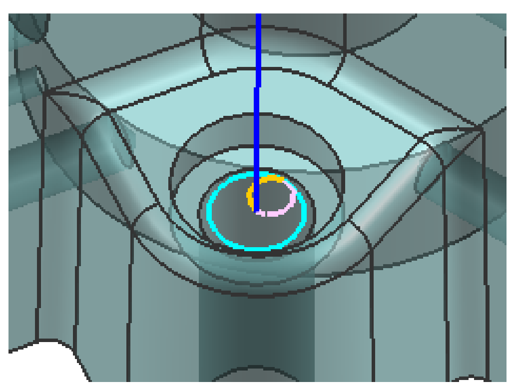
Mill hole chamfers using the **Hole Chamfer Milling** operation to cut a single chamfer on multiple features.

- Select hole features to chamfer, or select geometry from the graphics window. You do not need a modeled chamfer.
- For modeled chamfers, select chamfer machining areas to chamfer, or select geometry from the graphics window. For unmodeled chamfers, select the cylindrical face to chamfer and specify the counter sink diameter.
- When the angle of the cutter tip does not match the angle of the modeled chamfer:
    - The operation leaves material if the cutter angle is less than the chamfer angle.
    - The tool gouges if the cutter angle is greater than the chamfer angle. NX reports the gouges.

The operation determines the in-process feature volume for the chamfer and creates a single circular cut around the hole diameter.



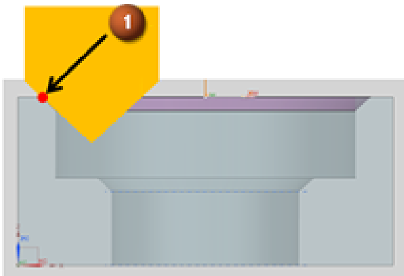
Chamfer in-process feature volume



Chamfer tool path

Tracking points

NX drives the tracking point you select as the drive point along the top edge of the chamfer. Usually the tracking point is located on the cutting edge, but NX can use any tracking point that you select.



1 Tracking point on spot drilling tool used as the drive point in the operation

> **Note**
>
> Because the operation cuts chamfers on holes with or without a modeled chamfer, NX uses the selected tracking point even if the tool gouges the part or cuts air.

Contact and tracking data output

NX optionally outputs contact/tracking data at the tracking point defined on the tool diameter. Otherwise, NX outputs the tool path at the tool tip centerline.

Where do I find it?

Application	Manufacturing
Prerequisite	A supported chamfering tool such as a chamfer mill, spot drill, or countersink tool. The tool must have the following attribute: • Defined tracking point.
Command Finder	**Create Operation**
Location in dialog box	Hole Chamfer milling operation **Create Operation** dialog box→**Type** list→**hole_making**→**Hole Chamfer Milling**  Selecting the tracking point **Hole Chamfer Milling** dialog box→**Path Settings** group→**Drive Point** list

Gouge reports for authorized penetrations

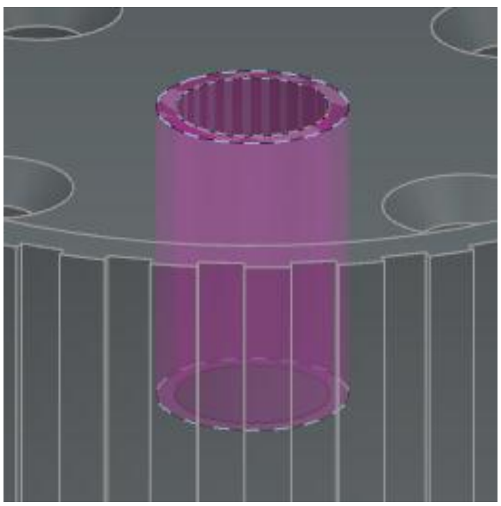
When a modeled hole is the tap drill size, you now can obtain a tool path for tapping operations without turning off gouge checking.

When you generate a tapping operation with the **Gouge Checking** option turned on, NX does not detect a gouge in the threaded volume. Other gouges are still detected and reported.

The **Gouge Check** command and the gouge checking options in verification and simulation no longer report a problem as long as:

- The hole in the part is larger than the minor diameter of the specified thread feature.
- The tool diameter is less than the major diameter of the thread feature.

The threaded volume that is used for gouge checking is the in-process feature for the hole, as shown.



After a gouge check of the tapping operation in the part shown, the **Information** window contains the following output:

```
Operation TAPPING: No gouged motions found.
Hybrid Gouge Checker used.
```

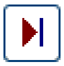
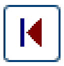
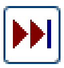
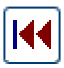
Where do I find it?

Application	Manufacturing
Prerequisite	Feature volumes must be defined for the operation.
Operation Navigator	Right-click operation→**Gouge Check** Right-click operation→**Tool Path**→**Verify**→**Replay**→**Gouge and Collision Settings**
Operation dialog box	**Generate** **Verify**→**Replay**→**Gouge and Collision Settings**
Tool Path Editor dialog box	**Analyze Tool Path** group→**Gouge and Collision Check**
Simulation Control Panel dialog box	**Simulation Settings** group→**Simulation Settings** →**Simulation Settings** dialog box→**Collision Detection** group→**Specify Collision Pairs** →**Specify Collision Pairs** dialog box→specify the part and the tool

Verification settings and playback enhancements

Keyboard shortcuts

When a playback button, for example **Step**, is selected, you can use keyboard keys to control the animation.

 **Step**	Right arrow key
 **Step Backward**	Left arrow key
 **Forward to Next Operation**	Page Up key
 **Rewind to Previous Operation**	Page Down key

For continuous motion, hold the corresponding arrow key.

Note

You can also use the arrow keys to move a selected slider or scroll a selected list.

Tool Path Visualization settings

The **Tool Path Visualization** dialog box settings are now retained throughout the NX session, with the following exceptions:

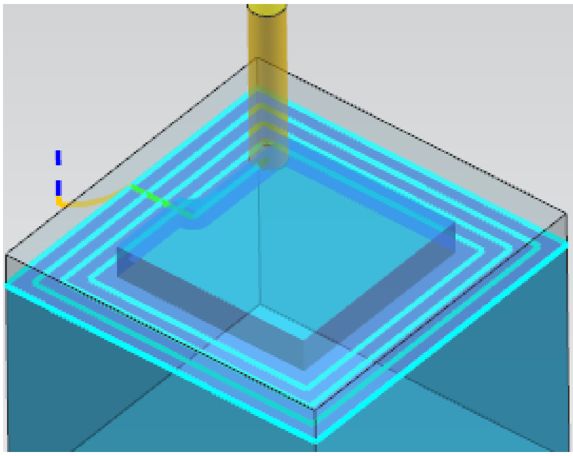
- The slider value that controls the listing in the path listing window is not retained.
- The **Replay** tab remains the default tab.

Note

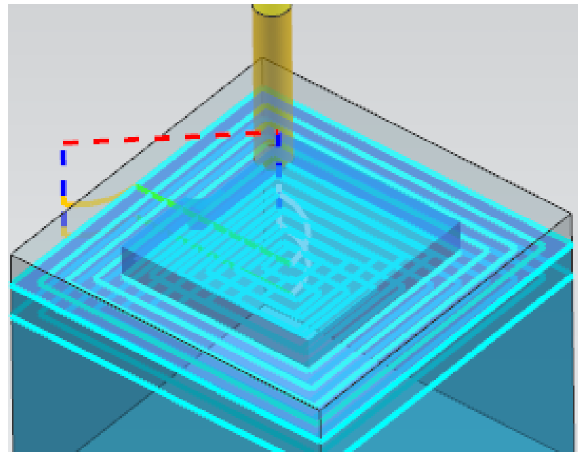
NX does not retain the settings in dialog boxes that you open from within the **Tool Path Visualization** dialog box.

Tool path display during playback

The **Tool Path** display options include a **Start to Current Motion** option, to display the path from the start point of the operation until the current position of the tool.



Start to Current Motion



All

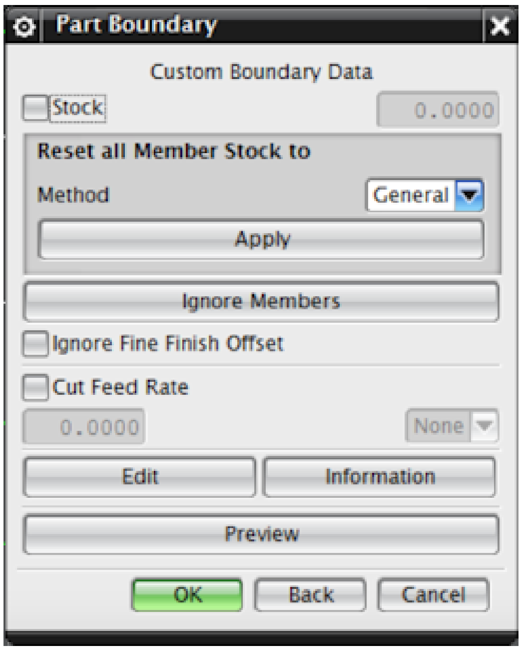
Where do I find it?

Application	Manufacturing
Prerequisite	You must have at least one operation that has been generated. For arrow key shortcuts, click a playback button to give it the focus. For settings retention, reopen the **Tool Path Verification** dialog box.
Location in dialog box	**Replay** or **3D Dynamic** tab→**Tool Path** list→**Start to Current Motion**

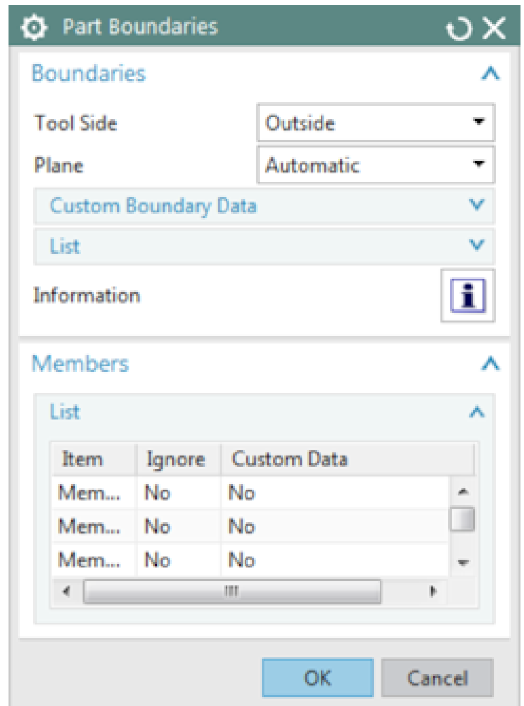
Boundary dialog box enhancements

The look of the dialog boxes where you select boundary options in NX turning has changed.

Pre-NX 9.0.2



NX 9.0.2



Turning workpiece

The names of boundary selection dialog boxes that you access from the **Turn Bnd** dialog box have changed. To display the **Turn Bnd** dialog box:

- In the Geometry view of the **Operation Navigator**, right-click **Turning Workpiece** and choose **Edit**.

Path to boundary selection dialog box	Pre-NX 9.0.2 name	NX 9.0.2 name
Click **Specify Part Boundaries** 	**Part Boundary**	**Part Boundaries**
Click **Specify Blank Boundaries** 	**Select Blank**	**Blank Boundaries**

Turning operation

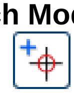
The name of boundary selection dialog boxes that you access from a turning operation have changed.

- Right-click any turning operation and choose **Edit**.

Path to boundary selection dialog box	Pre-NX 9.0.2 name	NX 9.0.2 name
Geometry group→**Custom Part Boundary Data**	**Part Boundary**	**Part Boundaries**

Teach Mode operation

The names of boundary selection dialog boxes that you access from a **Profile Move** type of **Teach Mode** operation have changed.

1. Create a **Teach Mode** operation.
2. In the **Teach Mode** dialog box, in the **Sub-Operations** group, click **Add New Sub-Operation** .
3. In the **Create Teachmode Subop** dialog box, from the **Move Type** list, select **Profile Move**.

Path to boundary selection dialog box	Pre-NX 9.0.2 name	NX 9.0.2 name
Path Settings group→**Drive Geometry** list→**New Drive Curve**	**Select Drive Geometry**	**Part Boundaries**
Start and Stop group→**Start Position** list→**Check Curve**→**Start Check Geometry**	**Select Start Check Geometry**	**Start Check Boundary**
Start and Stop group→**Start Position** list→**Check Curve**→**Stop Check Geometry**	**Select Stop Check Geometry**	**End Check Boundary**

Finishing enhancements for Curve/Point drive method

You can do the following using the **Curve/Point** drive method.

- Position the tool to follow along edges of the part geometry at a specified offset using the **Offset Left** option.
- Shift the tool contact point a distance along the curve tangent when using a non-center cutting tool such as a bullnose endmill. When cutting around an axis, this is equivalent to a rotational shift around the axis.
- Chamfer.

These enhancements are primarily intended to improve the surface finish for 5-axis machining of rotary floors. However, the **Curve/Point** drive method is not used exclusively for rotary floors, and the enhancements may be useful for other types of applications. When the tool axis is not perpendicular to the cut area, you may see differences in the 3-axis tool path for the **Offset Left** option. See the *Manufacturing product notes* section in the *Release Notes* for details.

Following edges at an offset

NX does the following to create the tool path:

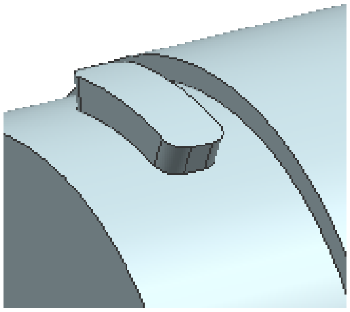
1. Offsets the selected edges or curves along the cut area surface to create the drive curves.
2. Checks for gouges against part or check geometry, and removes any portions of the tool path that gouge. If the offset value is too small to clear the geometry, NX does not create a tool path.

Note

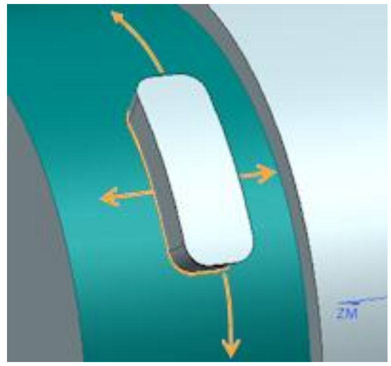
To specify an offset to the right, enter a negative value.

In the following example, the drive curves are selected from the feature edges.

Part to machine



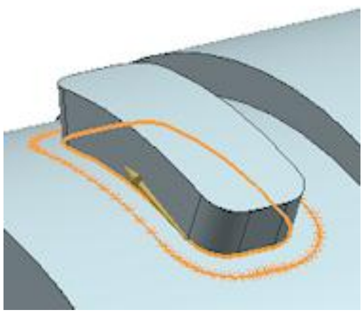
Cut area and edges to offset



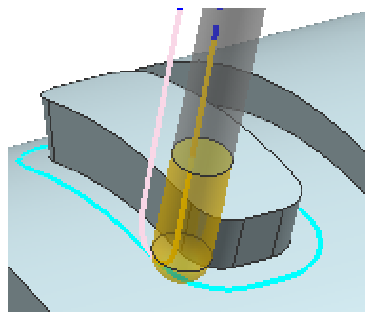
Tool diameter = 3 mm

Offset Left = 1.5 mm

Drive curves

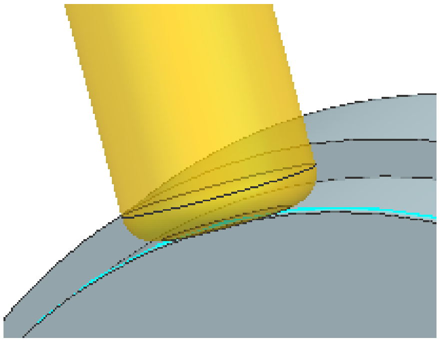
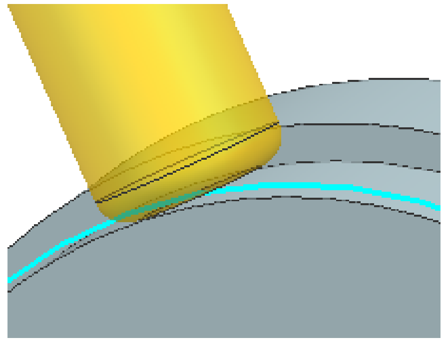


Tool path



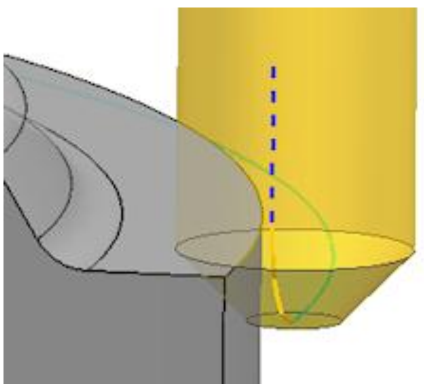
Shifting the contact

The following example shows the effect of shifting the tool contact by 1.5 mm on a 3 mm diameter tool.

Tool at default contact point	Tool contact shifted
	

Chamfering with Curve/Point

Use a chamfering tool and specify negative stock to cut a chamfer. When you generate the operation NX displays a warning message. To continue generating the operation, click **No** in the message box.



Why should I use it?

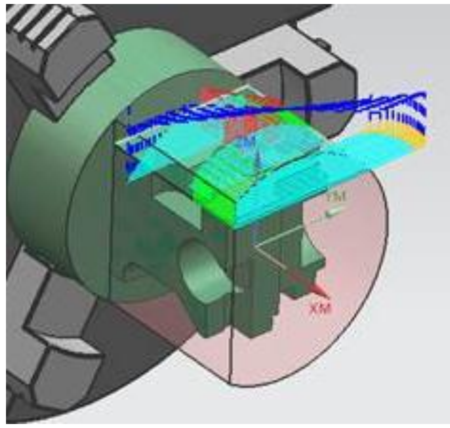
These enhancements help improve surface finish quality. Previously, there was no way to shift the tool contact, and it was necessary to manually create offset curves. You no longer need to create offset curves to drive the tool along part features.

Where do I find it?

Application	Manufacturing
Prerequisite	**Curve/Point** drive method
Location in dialog box	**Curve/Point Drive Method** dialog box→**Drive Settings** group→**Offset Left** and **Tool Contact Shift**

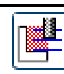
Performance enhancements for Floor Wall operations using a 3D IPW

What is it?



The time required to generate tool paths, or to edit an operation with the **Preview** option selected is now improved. The performance improvements depend on the complexity of the blank and 3D IPW, and are most noticeable when the operation uses a complex or nonrectangular blank.

Where do I find it?

Application	Manufacturing
Command Finder	**Floor and Wall with IPW** 

Support for MoriAPT CLSF output

NX can now output the CLSF for single channel Mori Seiki machine tools in the MoriAPT format developed by Mori Seiki. DMG/Mori Seiki provides a controller-based postprocessor and simulation package for their CNC machines.

When you output your machining data in the MoriAPT format, you can cut parts on Mori Seiki machines without having to create a postprocessor or postprocess the data in NX.

The MoriAPT format is an extension to the ISO4343 APT language format.

You must use the **CLSF Output** command, choose the **MORIAPT** format, choose a name and location for the file, and create the output by clicking **OK** or **Apply**.

> **Note**
>
> Because there is no NC code file or postprocessor, you can simulate only the tool path in NX.

Where do I find it?

Application	Manufacturing
Menu	**Menu**→**Tools**→**Operation Navigator**→**Output**→**CLSF**
Ribbon bar	[Select Operation or program group]→**Home** tab→**Operation** group→**More**→**Output CLSF**
Location in dialog box	**CLSF Format** list→**MORIAPT**

Chapter 3: Advanced Simulation

Importing .layup files

What is it?

You can now import ply definitions that are saved in the .layup format.

Where do I find it?

Application	Advanced Simulation
Command Finder	**Import Global Layup**
Location in dialog box	**File Type** list→**Layup files (.layup)**

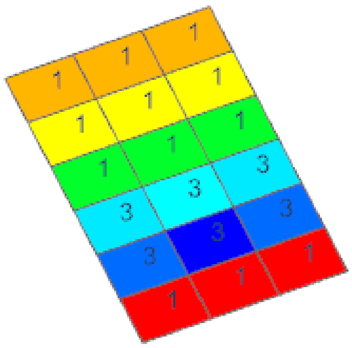
Show Critical Layer ID

What is it?

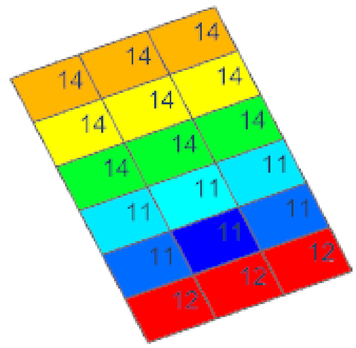
Use the **Show Critical Layer ID** command to display the ID of the critical layer for each element being displayed in the laminate graphical report of ply-enveloped results. The ply layer ID indicates the stacking order of the ply on the element.

Example

In this example, the global ply IDs are 11, 12, 13, 14, 15, and 16, but, each element contains only three plies.



Show Critical Layer ID



Show Critical Ply ID

For the top most elements and the bottom most elements, the critical layer is the first ply indicated by ID 1. For the top most elements, the first ply has a global ply ID of 14. For the bottom most elements, the first ply has a global ply ID of 12.

The Nastran PCOMP format supports layer IDs, as opposed to the PCOMPG format, which supports global ply IDs. Regardless of which Nastran format you chose to solve your simulation in, you can display the critical ply or layer in laminate post reporting.

Why should I use it?

The **Show Critical Layer ID** command lets you visually identify which layer is critical for each element of the model, independent of the ply global IDs. It is useful for graphical reports when you set the **Ply Export Option** list to **Layer**.

Where do I find it?

Application	Advanced Simulation
Prerequisite	The graphical report result sets must include envelope result sets.
Post Processing Navigator	Select an enveloping result set→right-click the **Post View** node→**Show Critical Layer ID**

Chapter 4: Shipbuilding

Assign names to basic design objects

What is it?

You can assign a name and a context attribute when you create a plate system, profile system, or seam in the Ship Structure Basic Design application.

- The dialog boxes for the plate system, profile system, and seam commands include a new group to specify the name and attribute.
- You can also assign names to the subsystems that result from splitting a plate system or profile system with a seam by editing the subsystem feature in the **Part Navigator**. When the basic design is transitioned to the detail design, the subsystem names are given to the newly created items or design elements.
- You can customize the naming rules for each object type in an XML file located at *%NXSHIP_DIR%\data\ShipNamesRecipes.xml*. The rules specify the field types that appear in the dialog box to format the name.

  For example, the following fields could be specified for a longitudinal bulkhead.

Field	**Value**	Description
Fixed	LBH–	A prefix that cannot be changed.
Option	P	P (port) or S (starboard). The default value is P.
Any	–	Any value specified. The default value is a dash.
Index	nnnn	A four digit index number that is automatically assigned.

  The possible names assigned to the bulkhead could be *LBH–P-0001*, *LBH–S-0002*, or *LBH–P_x0003*.

- You can define the allowed values for a context attribute in an XML file located at *%NXSHIP_DIR%\data\NX_ShipAttribute.xml*. When you create the feature, you can select one of the values from a menu in the dialog box.

  For example, you could specify the context attribute value for a pillar system as *Pillar*, *Beam*, *Stanchion*, *Support*, *Horizontal Brace*, *Foundation Member*, *Platform Member*, or *Post*.

Why should I use it?

Assign custom names and context attributes to help describe objects in the basic design. The names and attributes can be applied in downstream manufacturing and drafting processes.

Where do I find it?

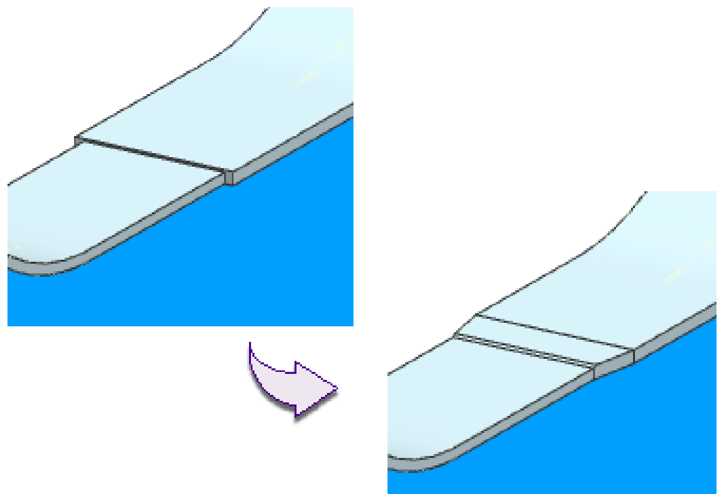
Application	Ship Structure Basic Design
Command Finder	**Hull** **Deck** **Transverse Bulkhead** **Longitudinal Bulkhead** **Stiffener System** **Edge Reinforcement System** **Seam** **Pillar System**
Location in dialog box	**Names** group specific to each command. For example, the **Deck** dialog box includes a new **Deck Names** group.

Profile Transition enhancements

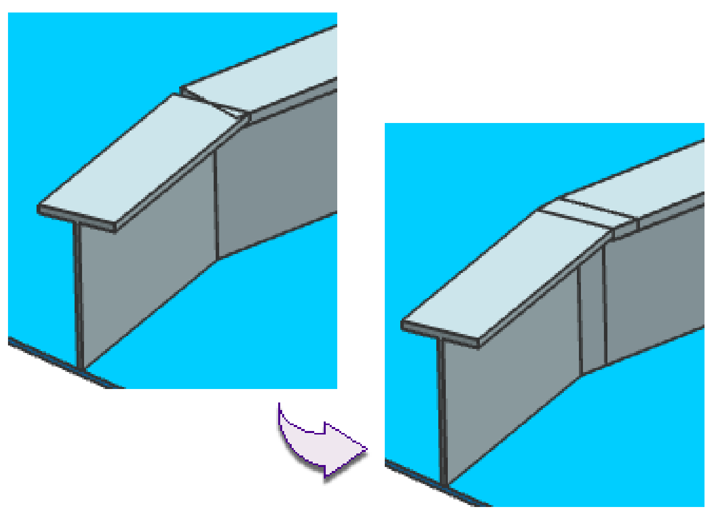
What is it?

The **Profile Transition** command is enhanced so that you can:

- Create transitions between edge reinforcements. The opposing end cuts must be in the same plane and the up directions must be parallel. A chamfer may be automatically applied when the target thickness is greater than the boundary thickness.



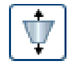
- Create transitions between some misaligned profiles. You can select profiles that have end cuts in different planes.



Why should I use it?

These enhancements save time by automatically creating the transitions between different size edge reinforcements and between stiffeners that are not aligned.

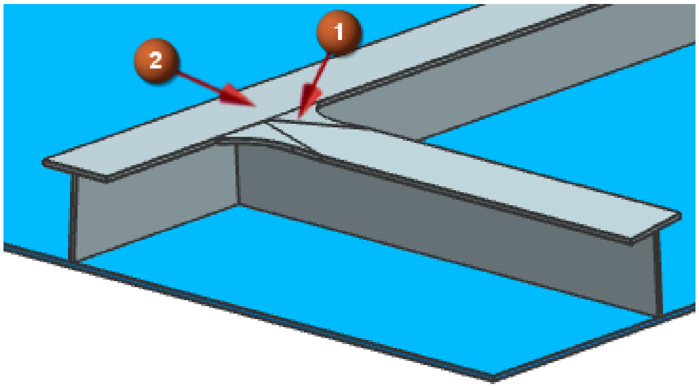
Where do I find it?

Application	Ship Structure Detail Design
Command Finder	**Profile Transition** 
Location in dialog box	**Parameters** group→**Misaligned Transitions**

Add knuckles to flared end cuts

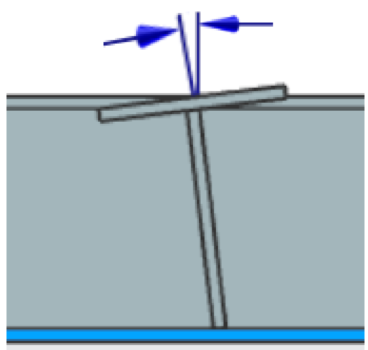
What is it?

The **End Cut** command is enhanced so that you can add knuckles to flared end cuts on edge reinforcements and built-up profiles (1) so that they can be bent to match a boundary (2).



To create a knuckled end cut:

- The end cut must be applied to an edge reinforcement or the flange of a built-up stiffener.
- The **Limit Type** must be set to **Neat Trim**.
- The boundary object must be a solid body.
- The top face of the boundary must touch the flared plate.
- The angle between the normal of the top face of the boundary and the top face of the flared plate must be greater than the knuckle angle clearance. The clearance is defined in the KNUCKLE_ANGLE_CLEARANCE column of the parameter spreadsheet for the flared end cut section type.

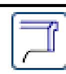


- The intersection of the top face of the boundary and the flared face must be near the intersection of the mold plane and the flared face. You can specify a value for the **Knuckle Distance Tolerance** in the customer defaults.

Why should I use it?

Flared end cuts produce wider end plates that are welded to a selected boundary. Adding knuckles lets you bend the end plates to match the boundary so that they are easier to weld.

Where do I find it?

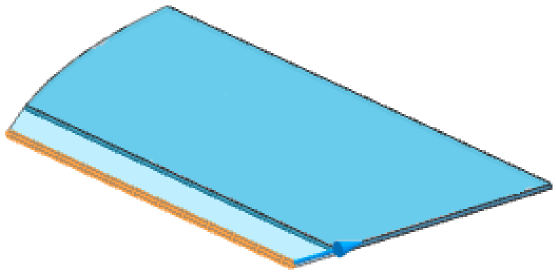
Application	Ship Structure Detail Design
Command Finder	**End Cut** 
Location in dialog box	**End Treatment** group→**Select Flange Boundary**

Excess Material enhancements

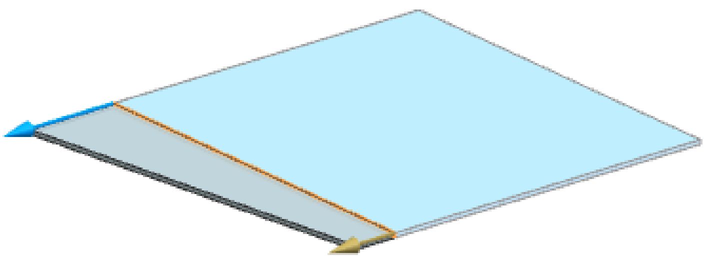
What is it?

The **Excess Material** command is enhanced so that you can:

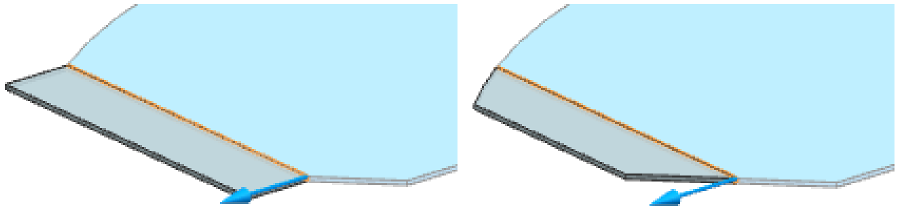
- Add excess material to multiple plates at one time
- Use the new **Fit Up** option to remove material from a plate.



- Use the new **Linear Varying** offset option to specify different offset values at the start and end of the selected face.



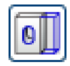
- Select options to extend the side faces when you add material to plates with corner cuts or edge cuts.



Why should I use it?

You can use additional methods to add or remove material from the ends of plates for manufacturing. You can also save time by selecting faces from multiple plates at once.

Where do I find it?

Application	Ship Structure Manufacturing
Prerequisite	Define cutting side faces in plates using the **Cutting Side Face** command.
Command Finder	**Excess Material** 
Location in dialog box	**Type** group→**Fit Up** **Offset** group→**Offset Type**→**Linearly Varying** **Settings** group→**Extend Face at Start** or **Extend Face at End**

Manufacturing XML Output enhancements

What is it?

The **Manufacturing XML Output** command is enhanced so that you can create and display the following types of markings:

- Knuckle curves
- Edge preparation
- Weld bevel change
- Edge radius
- Stiffener alignment marks at profile cutouts

The dialog box includes a new **Validation** group to let you display the plate markings before outputting XML information. The XML output also includes welding edge preparation parameters and the adjusted part boundary resulting from the edge preparation.

Why should I use it?

These enhancements let you preview and validate plate markings before outputting the information to an XML file.

Where do I find it?

Application	Ship Structure Manufacturing
Command Finder	**Manufacturing XML Output** 

Profile Sketch enhancements

What is it?

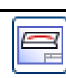
The **Profile Sketch** command includes the following enhancements:

- Profiles with built-up section types are supported.
- Edge reinforcements created using the **Stiffener/Edge Reinforcement** command are supported.
- Pillars are supported.
- New end cut types introduced in NX 9.0 are supported. If all dimensions for an end cut type use default values, only the type code is shown in the drawing.
- The marking side is shown as the main view on the drawing.
- There are now two options to determine the pitch for an inverse bending line: frame and value. When the inverse bending line crosses three and more frames, the frame option is used. Otherwise, the value option is used.
- The profile length in the drawing is calculated before end cuts are added.
- Mounting angles are supported. If a stiffener has a mounting angle, a symbol is included on the drawing.
- Show frame lines on the drawing if the stiffener is not bent. Dimensions to water stops are measured from the nearest frame line.
- Add a second direction to the Direction Index symbol when the profile section is symmetrical in one direction. For example, this can apply to profiles with round, square, I type, or H type sections.
- Add the correct value in the bend method column on the drawing (HEAT or PRESS).
- Include the major drain or air hole name in the **Hole** column on the drawing.

Why should I use it?

These enhancements let you create drawings for additional types of profiles that satisfy requirements at your shipyard.

Where do I find it?

Application	Drafting
Prerequisite	Open an existing assembly or workset containing stiffeners, edge reinforcements, or pillars. If the assembly or workset contains bent stiffeners, use the **Inverse Bending Lines** command in the Ship Structure Manufacturing application to create the inverse bending lines.
Command Finder	**Profile Sketch** 

Specifying locations of the configuration files:

Menu	**File→Utilities→Customer Defaults**
Location in dialog box	**Ship Drafting→Profile Sketch**

Chapter 5: Tooling Design

Morphing objects using OmniCAD

What is it?

You can morph objects in the NX CAD environment using the following integrated OmniCAD commands.

 **OmniFree Transformer**

Use this command to morph objects using points and curves as constraints.

 **OmniMesh Transformer**

Use this command to morph objects using CAE Meshes, STL files, or point clouds.

Note

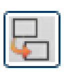
Siemens PLM Software recommends that you use an NX facet body instead of a JT facet body.

 **OmniCopy Transformer**

Use this command to copy the parameters of the transformed objects and then transform the objects by using the copied parameters.

 **Flow Blend**

Use this command to create constant or variable blends along complex faces that have small curvature. These blends are easy to manufacture and increase product durability and aesthetic appeal.

 **OmniSwitch Application**

Use this command to export NX objects that you want to transform into the OmniCAD software.

 **Save OCP File**

Use this command to export NX objects that you want to transform into the selected OmniCAD part file and then save the selected OmniCAD part file.

Note

Make sure that the environment variable `$OmniCAD_NX90_DIR` is available in the `\ugii\menus\custom_dirs.dat` file. If this environment variable is not available, copy the `\ugii\menus\custom_dirs.dat` file to your local drive, add the environment variable `$OmniCAD_NX90_DIR` to it, and then set the environment variable `UGII_CUSTOM_DIRECTORY_FILE` to `<local drive path>\custom_dirs.dat`

Where do I find it?

Menu	**OmniCAD For NX**→[OmniCAD commands]

Chapter 6: CMM Inspection Programming

General enhancements

Overriding dimensional tolerance values defined in PMI

Dimensional tolerance values defined in a master model may not reflect the design intent. For example, an acceptable angle on the part may be 89.998 degrees instead of the 90 degrees prescribed in a drawing, with acceptable measurement variation causing the tolerance to fail. In the **Tolerance Operation** dialog box, in the **Parameters** group, you can override dimensional **Nominal**, **Upper**, and **Lower** tolerance values linked to PMI.

When you override these values, the **Inherited** icon changes to a **Locally defined** icon to indicate that the PMI value has been overridden. To recall the PMI value, click the unlock icon and choose **Inherited from PMI**.

Collision avoidance progress bar

When you run collision avoidance on inspection paths, a progress bar reports the percentage complete and displays each action.

Performing analysis... 28% **PATH_CYLINDER_TOP - Find Sensor**

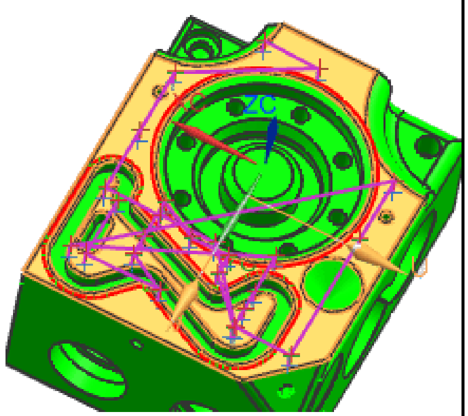
Where do I find it?

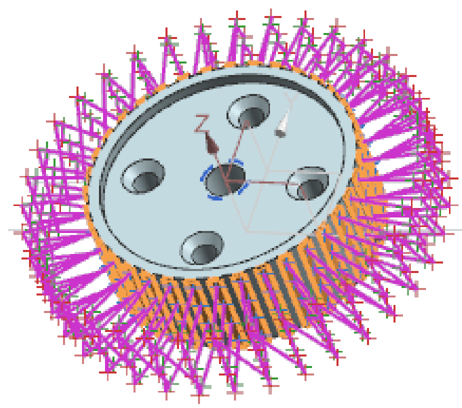
Application	CMM Inspection Programming
Prerequisite	• To define a dimensional tolerance operation, create an appropriate inspection feature. • To apply collision avoidance, in the **Inspection Navigator**, select one or more inspection paths.
Command Finder	**Collision Avoidance**

Creating plane and cylinder features from multiple faces

What is it?

You can create a single plane feature from multiple coplanar faces, and a single cylinder feature from multiple coaxial surfaces that together form a partial cylinder. When you define a point set sub-operation for either feature in the **Inspection Path** dialog box, each face gets a separate UV grid.





The same logic applies if a tolerance is linked to multiple coplanar faces or coaxial surfaces in PMI. When you link to PMI, the faces or surfaces are generated as single features, and a single inspection path is applied.

Why should I use it?

When you create single plane and cylinder inspection features from multiple faces, you avoid the need to create multiple general surface features and individual inspection paths for those features.

Where do I find it?

Application	CMM Inspection Programming
Command Finder	Plane, Cylinder
Location in dialog box	Creating a single plane feature from multiple coplanar faces: **Plane Inspection Feature** dialog box→**Geometry** group→**Select Plane or Base Point** Creating a single cylinder feature from multiple coaxial faces: **Cylinder Inspection Feature** dialog box→**Geometry** group→**Select Cylinder or Base Point**

Specifying user-defined events (UDEs)

What is it?

You can specify user-defined events (UDEs) to create dialog boxes that let you quickly insert custom code into your post-processed DMIS inspection programs. If you use UDEs, you do not have to write the same code in each program.

When you post process your program, the manufacturing output manager (MOM):

1. Looks for event parameters generated from UDE dialog boxes. Dialog box parameters are defined in a *ude.cdl* file.
2. Handles these event parameters to output custom DMIS code. Event handlers are defined in a *ude.tcl* file.

Why should I use it?

A custom code event might include a command that CMM Inspection Programming does not support, or a complex function that otherwise takes too much work to define for every inspection program. UDEs make custom code reusable.

Where do I find it?

Application	CMM Inspection Programming
Prerequisite	Define your dialog boxes in the and event handlers in the *ude.cdl* file and event handlers in the *ude.tcl* file, using a text editor. Restart NX to insert them into your inspection programs.
Inspection Navigator	Right-click the object before which you want the event to occur→**Object**→**Start Events** or **End Events**

Chapter 7: Data translation

Support for ESKD standard entities using DXF/DWG and 2D Exchange translators

When you use the DXF/DWG or 2D Exchange translators, you can now export the following:

- ESKD section lines
- ESKD symbols
- Limits and fits tolerance types
- Dimensions with the **text above the stub** style
- Viewing direction reference arrow

Where do I find it?

Command Finder	Export AutoCAD DXF/DWG File Export 2D Exchange File

CATIA V5 translator enhancement

You can now export and import wireframe color, font, and line width between CATIA V5 and NX.

Where do I find it?

NX CATIA V5 Interface

Command Finder	CATIA V5

NX Data Exchange CATIA V5 Interface

Menu	**Siemens NX x.x→Translators→CATIA V5→CATIA V5 Import** **Siemens NX x.x→Translators→CATIA V5→CATIA V5 Export**

Chapter 8: Routing

Convert Splines

What is it?

Use this command to convert pre-NX 9.0 splines in the work part to new splines. NX no longer uses the pre-NX 9.0 splines.

Why should I use it?

Each time you edit a pre-NX 9.0 spline, NX converts it to a new spline. If you use this command before you start working, the conversion process will not slow you down when you edit splines.

Where do I find it?

Application	Routing
Command Finder	**Convert Splines** 

Cut elbow placement enhancements

What is it?

You can now place standard and cut elbows in a single operation using either of the following methods:

- Drag an elbow from the **Reuse Library**, and drop it on a corner RCP in the graphics window.
- Use the **Place Part** command.

  When you use the **Place Part** command, you can also select an elbow that is already used in the assembly to place standard or cut elbows on other corners.

To use these methods, you must set the `ALLOW_CUT_ELBOW` attribute to `TRUE` in the PTB file of the family of elbows.

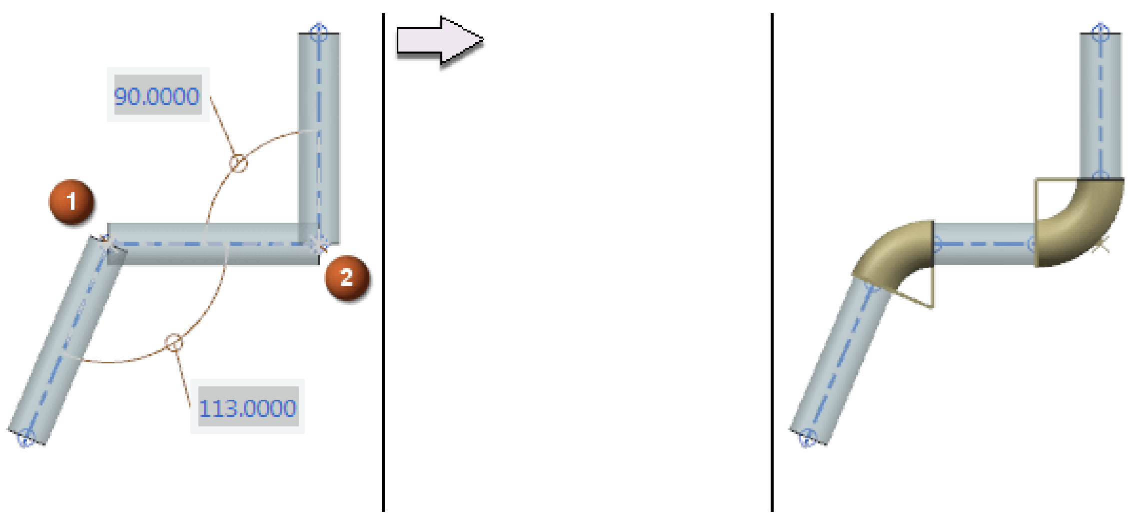
When you use either of the methods, NX identifies the angle at which the two segments form the intersection, calculates the required elbow angle, cuts the elbow if required, and places it.

You can also place an elbow on the port of one of the two segments that form the intersection.

Example

Consider the following segments with two corners. Intersection (1) which is 113°, requires a cut elbow of 67°. Intersection (2) which is 90°, requires a standard elbow of 90°.

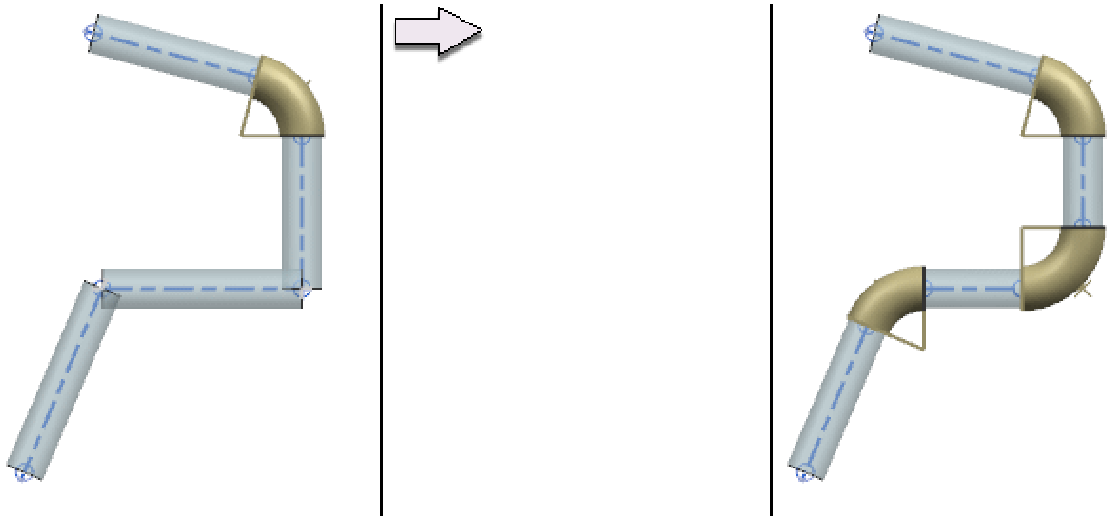
When you select a standard elbow of 90° from the **Reuse Library**, and place it at both the intersections, NX cuts the elbow to 67° while placing it at intersection (1), and places the elbow without cutting it at intersection (2).



Example

Consider the following segments with one existing cut elbow, and two intersections on which you want to place elbows.

If you use the **Place Part** command, you can select the existing cut elbow in the graphics window and use it to place elbows to complete the remaining corners. NX uses the underlying standard elbow part from the **Reuse Library** for the subsequent placement of elbows, irrespective of whether the corners require standard elbows or cut elbows.



Note

You can still place cut elbows using the **Cut Elbow** command or while using the **Create Linear Path** command with the **Allow Cut Elbow** check box selected.

Why should I use it?

The advantage of these new methods is that you can use them to place standard as well as cut elbows. This is useful in the following cases:

- Your design of segments is ready, and you have to assign elbows to multiple corners.
- Your design already has an elbow, and you want to assign the same elbow part to other corners in the network.

Where do I find it?

<table>
<tr><td>Application</td><td>Routing Mechanical</td></tr>
<tr><td>Prerequisite</td><td>To use these new methods, you must add the following attribute to the PTB file of the family of elbows or as a new column for a part in a PTB file.
<table>
<tr><td><code>APPLIED</code></td><td></td></tr>
<tr><td><code>NAME</code></td><td><code>ALLOW_CUT_ELBOW</code></td></tr>
<tr><td><code>FORMAT</code></td><td><code>TRUE</code></td></tr>
<tr><td><code>END_OF_APPLIED</code></td><td></td></tr>
</table>
</td></tr>
</table>

	Tip • Set the `ALLOW_CUT_ELBOW` attribute to `TRUE` for elbows such as butt weld elbows that you intend to use for cut elbows. For elbows such as flanged elbows and threaded elbows, set the attribute to `FALSE` in the respective PTB files to ensure that they are not cut. The attribute and its value are case-sensitive. • If you do not add this attribute to the PTB file, you can neither use the **Place Part** command nor drag standard elbows from the **Reuse Library** and drop them on the corner RCPs to cut and place them. The attribute and its value are case-sensitive. If you use lower case alphabets, NX behaves as if the `ALLOW_CUT_ELBOW` attribute does not exist in the PTB file.
Command Finder	**Place Part** 

Assign Corner enhancement

What is it?

When you use this command to create a corner the attributes of the corner object are inherited by the corner segment and corner RCP.

Where do I find it?

Application	Routing
Command Finder	**Assign Corner**

Unify Path enhancement

What is it?

NX stops the **Unify Path** command from unifying paths if the following routing objects are a part of your selection:

- Direct mount parts
- Eccentric segment

Now, NX stops the **Unify Path** command from unifying paths if the following additional routing objects are a part of your selection:

- Eccentric reducer parts
- Parts that are placed using the **Instance Name Lookup** command

Where do I find it?

<table>
<tr><td>Application</td><td>Routing</td></tr>
<tr><td>Prerequisite</td><td>You must add the following attribute to the PTB file of the family of eccentric reducer parts.
<table>
<tr><td><code>APPLIED</code></td><td></td></tr>
<tr><td><code>NAME</code></td><td><code>NX_BLOCK_UNIFY</code></td></tr>
<tr><td><code>FORMAT</code></td><td><code>TRUE</code></td></tr>
<tr><td><code>END_OF_APPLIED</code></td><td></td></tr>
</table>
The attribute and its value are case-sensitive.</td></tr>
<tr><td>Command Finder</td><td>Unify Path</td></tr>
</table>

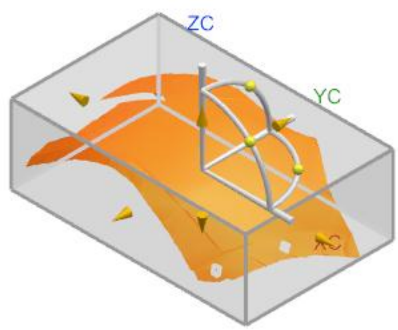
Chapter 9: Facet Body Preparation

Create Box

Use the **Create Box** command to create a bounding box around selected geometry.

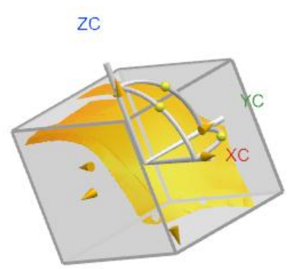
When you use the **Create Box** command:

- Curves, edges, faces and faceted bodies may be selected.
- The box is created to enclose the selected geometry.
- The box is aligned with the **WCS**.
- A clearance distance can be applied to the bounding box.
- An associative solid body is created.

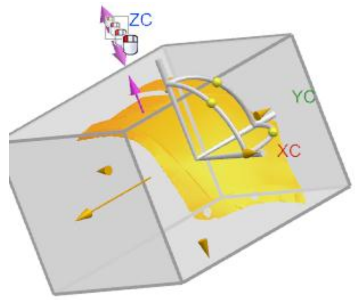


When you create a box, you can control:

- The reference CSYS orientation using the manipulator.



- The color of the bounding box.
- An offset from the body by using a clearance or by dragging the sizing handles on the box.

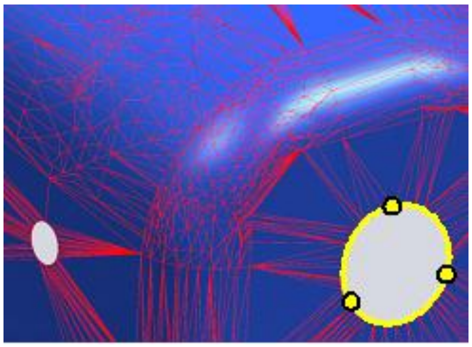


Where do I find it?

Application	Modeling, Mold Wizard, and Progressive Die Wizard
Prerequisite	A curve, edge, face, or a faceted body to select.
Command Finder	**Create Box**

Snap Point enhancement

A new **Point on Facet Vertex** option has been added to the snap point options, use this option to snap to facet vertices when creating lines, arcs, and circles based on a faceted body.



The **Point on Facet Vertex** snap point is currently available for the following commands:

- **Point**
- **Line**
- **Arc/Circle**
- **Lines and Arcs**
- **Studio Spline**
- **Fit Curve**
- **Four Point Surface**
- **Rapid Surfacing**

Where do I find it?

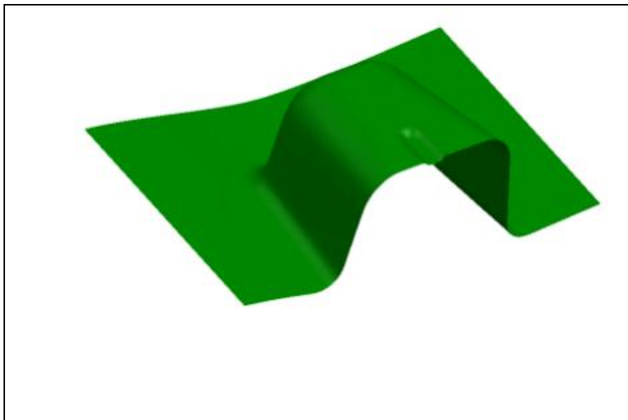
Application	Modeling
Top Border Bar	**Point on Facet Vertex**

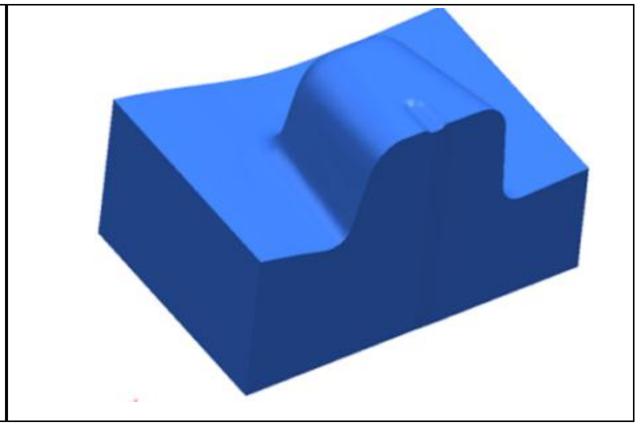
Extrude Facet Body

Use the **Extrude Facet Body** command to extrude an existing faceted body into a faceted solid body.

The faceted sheet:

- Can be extruded by distance.
- Can be extruded to a plane.
- Will process undercuts.





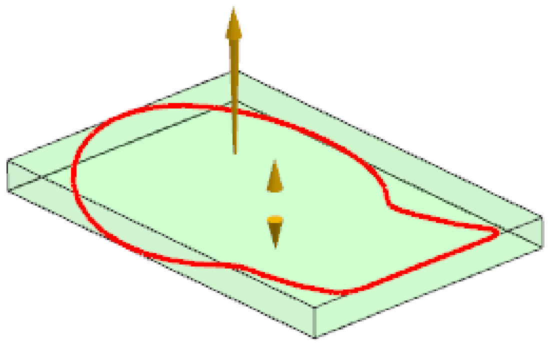
Where do I find it?

Application	Modeling
Prerequisite	A faceted body
Command Finder	**Extrude Facet Body**

Extrude Profile

Use the **Extrude Profile** command to extrude a closed profile between two planes.

- The curve is extruded along the specified vector and trimmed by limiting planes.
- The distance is measured from a bounding box of the input.
- An offset value may also be applied.
- A faceted body is created.



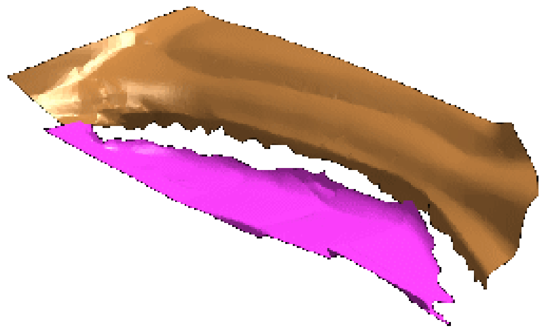
Where do I find it?

Application	Modeling
Command Finder	**Extrude Profile** 

Merge Disjoint Facet Bodies

Use the **Merge Disjoint Facet Bodies** command to bridge the gap between two disjointed faceted sheet bodies and to create a single merged faceted sheet body.

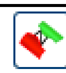
Faceted bodies may be joined using a linear or tangent option.



Why should I use it?

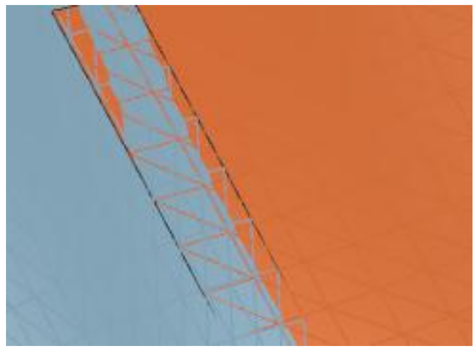
Faceted bodies sometimes have gaps between them that are not easily filled by additional scanning or by manual methods. **Merge Disjoint Facet Bodies** provides a convenient way to merge the bodies and automatically fill the gap between them.

Where do I find it?

Application	Modeling
Prerequisite	Two faceted bodies that are not touching
Command Finder	**Merge Disjoint Facet Bodies** 

Merge Overlapping Facet Bodies

Use the **Merge Overlapping Facet Bodies** command to merge two faceted bodies into a single merged faceted sheet body.



Why should I use it?

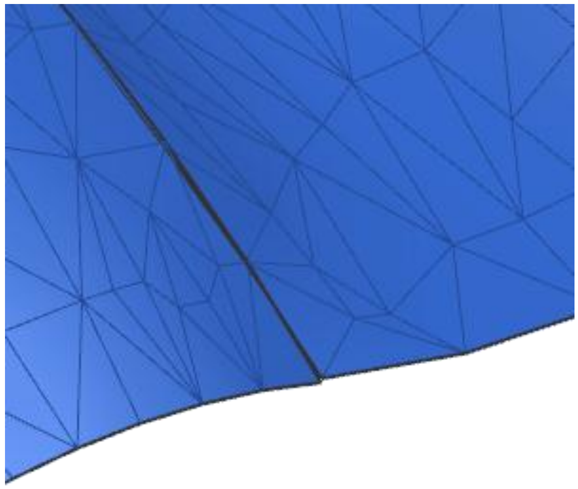
Faceted bodies sometimes overlap and the area in common needs to be combined in a way that does not produce an overabundance of facets. **Merge Overlapping Facet Bodies** provides a convenient way to merge the bodies and re-tessellate the common areas.

Where do I find it?

Application	Modeling
Prerequisite	Multiple faceted bodies
Command Finder	**Merge Overlapping Facet Bodies**

Merge Touching Facet Bodies

Use the **Merge Touching Facet Bodies** command to merge two faceted sheet bodies that are touching at a common edge by sewing them together to create a single new faceted sheet body.



Why should I use it?

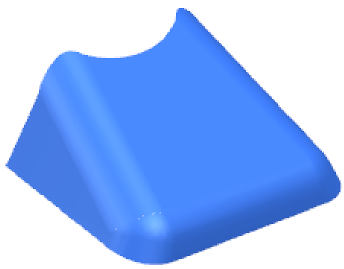
Even though two adjacent faceted sheet bodies may appear to touch at a common edge, there are often tiny gaps between them. **Merge Touching Facet Bodies** eliminates the gap and combines the bodies into a single faceted sheet body.

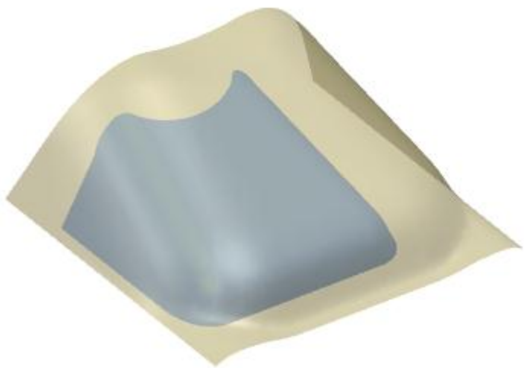
Where do I find it?

Application	Modeling
Prerequisite	Faceted sheets
Command Finder	**Merge Touching Facet Bodies**

Rough Offset enhancement

You can now use a facet sheet body as input for a **Rough Offset**.





Where do I find it?

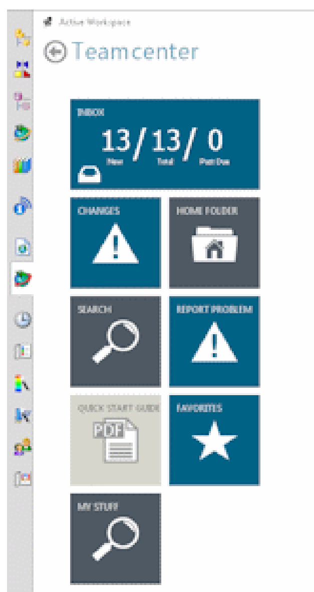
Application	Modeling
Command Finder	**Rough Offset**
Location in dialog box	**Filter** list **Surface Generation Method→Rough Fit**.

Chapter 10: Active Workspace

What is it?

Active Workspace is a Teamcenter web client that provides Teamcenter functionality in a wide range of browsers, including browsers on mobile devices. The Teamcenter **Active Workspace** web client is now incorporated into NX in addition to the NX **Teamcenter Navigator** and provides a seamless user experience between Teamcenter and NX.

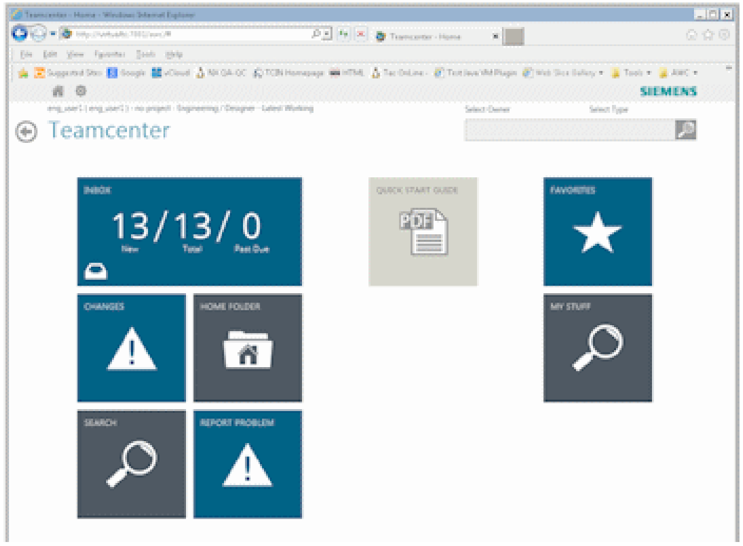
By incorporating **Active Workspace** inside NX, a similar user experience is provided whether you are using **Active Workspace** embedded in NX or as a web client. Currently, the functionality is similar to what you can do in the **My Teamcenter** application in Teamcenter, such as view and sift through items, item revisions, and datasets. However you can also create workflows, show related drawings, perform fast and comprehensive searches, and utilize change management. This is all packaged in a new interface that provides robust functionality.

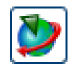


When you access **Active Workspace** from NX, you can:

- Select a part in **Active Workspace** and open it in NX.
- Select a part in **Active Workspace** and add it as a component to the work assembly in NX.
- Synchronize the selection of the parts between **Active Workspace** and NX. When you select a part in **Active Workspace** and the corresponding part is loaded in the NX session, then all of the occurrences of the part are selected. Similarly, if you select a part in NX and the part is visible in the current **Active Workspace** view, then it is automatically selected in **Active Workspace**.
- Select the part in NX and open the summary page for the selected part in **Active Workspace**.

In addition, from the **Active Workspace** web client, you can select a part and launch a new session of NX, which opens the selected part.



For this release, **Teamcenter Navigator**  and **Active Workspace**  coexist, and both are available from the NX Resource bar. In a future release, **Active Workspace** will replace **Teamcenter Navigator** as the data management tool in NX.

> **Note**
>
> **Active Workspace** is available only when you are running Teamcenter 4–tier deployment architecture. You must setup **Active Workspace** so that it is integrated with NX.
>
> For additional details on the full capabilities of **Active Workspace**, see the Teamcenter *Active Workspace Quick Start* guide.

Why should I use it?

With embedded **Active Workspace**, you have access to a broader range of PLM data and capabilities, such as workflow processes, change processes, documents, and so on.

Where do I find it?

Application	Teamcenter Integration for NX
Resource bar	**Active Workspace**