TEAMCENTER

Resource Manager on Active Workspace

Teamcenter 2412

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1. What is Resource Manager?

Resource Manager lets NC programmers, process planners, and tool designers store, modify, retrieve, and manage resource data to create:

- Fixture assemblies, machine setups, and NC programs for machine tools, robots, and measuring devices.
- Tool assemblies that combine machine tools and robots with their devices.
- Setup sheets for the complete operation.

Resource data can include classification information, Teamcenter items, and additional data such as associated imported files and links to context information. You define the data using hierarchically organized class definitions. Each class defines a set of values, or attributes, that typically specify characteristics of an object.

> Example:
>
> The diameter, weight, and length of a tool.

Use class hierarchies to organize and search for your resources as a hierarchically organized collection that is independent of product structure.

- Resource Management
  - Tools
    - Assemblies
      - ◊ Turning Tools
      - ◊ Milling Tools
      - ◊ Drilling Tools

Basic concepts

Use Resource Manager to:

- View resource components and their associated attributes using a variety of *views* or *perspectives*.
- Create detailed classification structures of manufacturing data such as tools, machines, fixtures, assemblies, and components.

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- Create structured resource assemblies used in your manufacturing tasks, propagate component attributes to your resource assembly, assign workflows to resources, and attach notes and graphics to resources.

- Classify and search for resource assemblies and resources using the Teamcenter quick search, **Object ID** search, or Classification Search Dialog.

- Create and edit manufacturing documentation to support rich work instructions and collaboration between departments.

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2. Getting started with Resource Manager

Selecting a workspace in Resource Manager

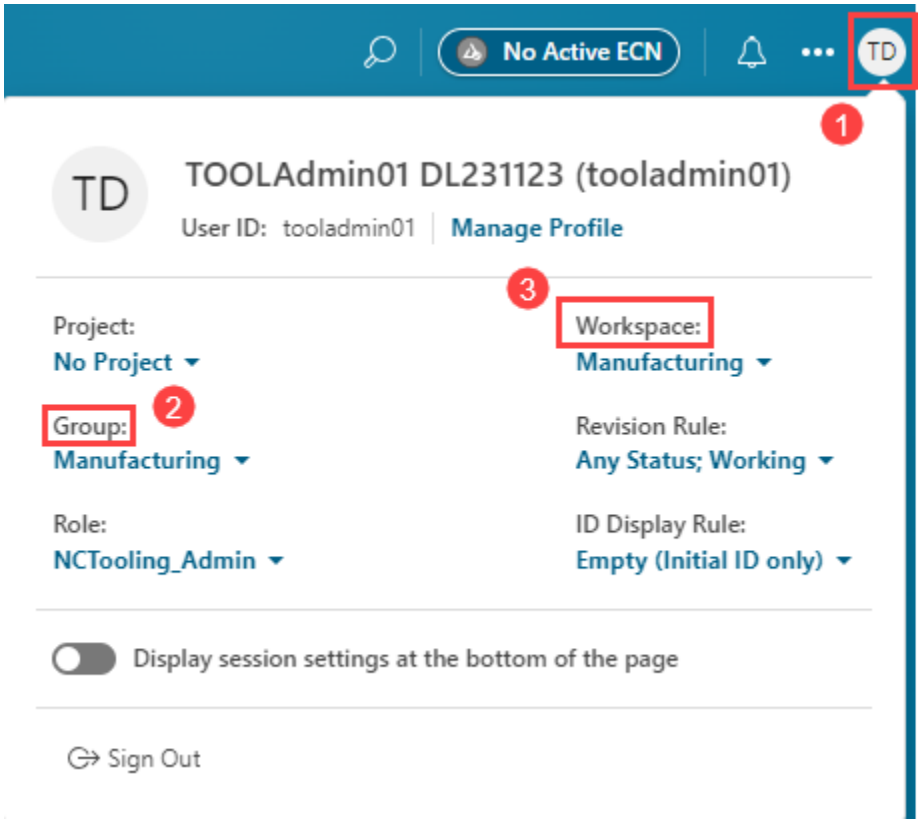
You can create multiple Resource Manager workspace profiles in Active Workspace so that only resource information and tiles that apply to a specified project are available for the user to use. This allows the user to quickly find resource information and tiles that are to be used for the specified project.

The default Resource Manager **Workspace** profiles in Active Workspace are:

Manufacturing: Allows the user to view and use resource information and tiles that are assigned to the project.

Author: Allows the user to view all of the resource information and tiles in the profile, regardless of the project.

You access the user attribute panel when you click the (1) user icon. Then select a group profile from the (2) **Group** menu or a workspace profile from the (3) **Workspace** menu.



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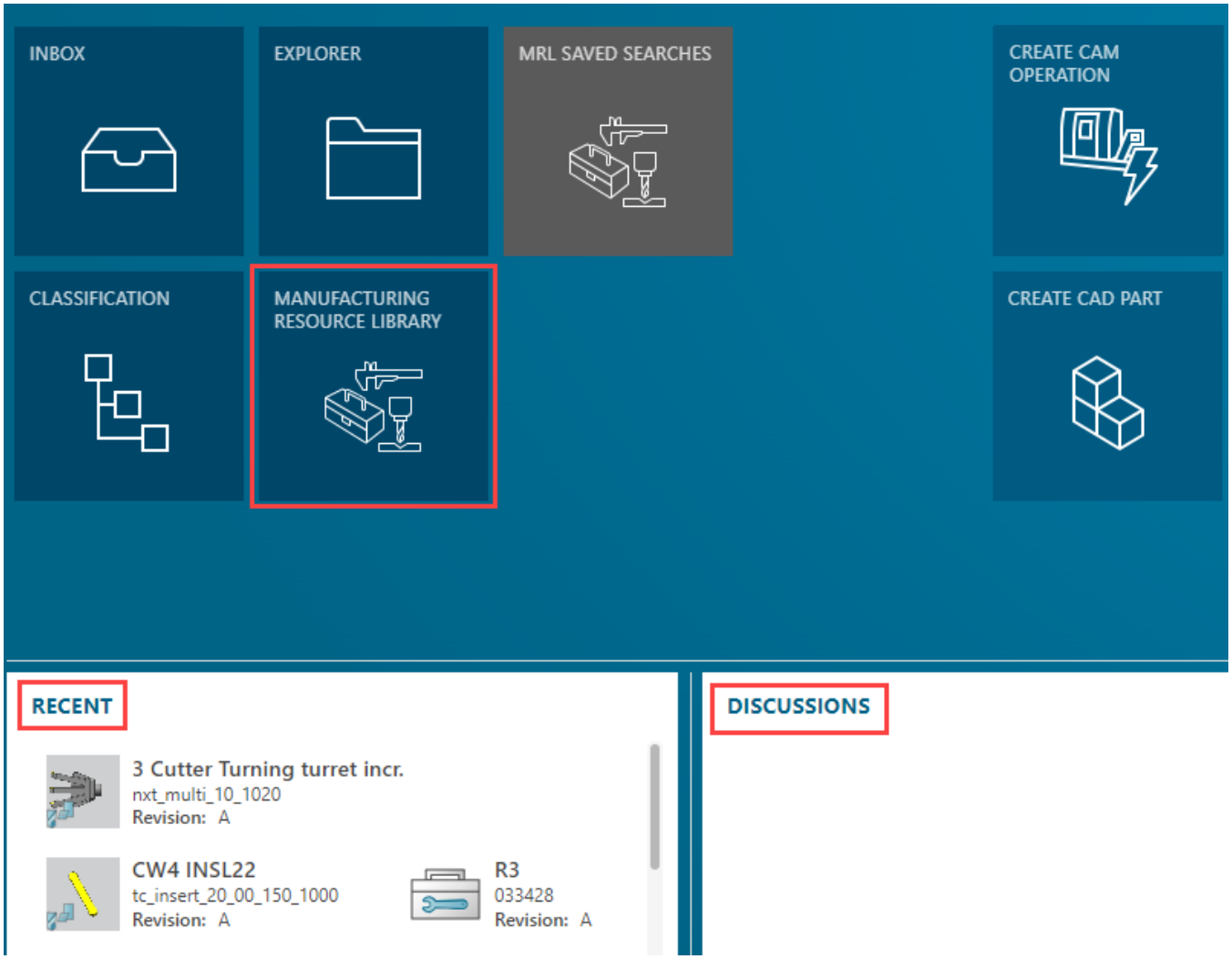
Manufacturing Resource Library dashboard

You can open the Manufacturing Resource Library (MRL) dashboard from the Active Workspace home page.



The MRL dashboard gives you access to information about the classified resources included in the MRL and their resource types.

> Note:
>
> Use the **RECENT** panel to quickly access and open previously viewed resources.
>
> You can use the **DISCUSSIONS** panel to communicate with other users.

Resources in the Manufacturing Resource Library main class include:

- **Factory Resources**

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- **Fixtures and Devices**
- **Machines**
- **Machining Data Library**
- **Measuring Devices**
- **New Resources**
- **Templates**
- **Tools**
- **Vendor Catalogs**

The dashboard has a bar graph that shows the number of resources by resource type.

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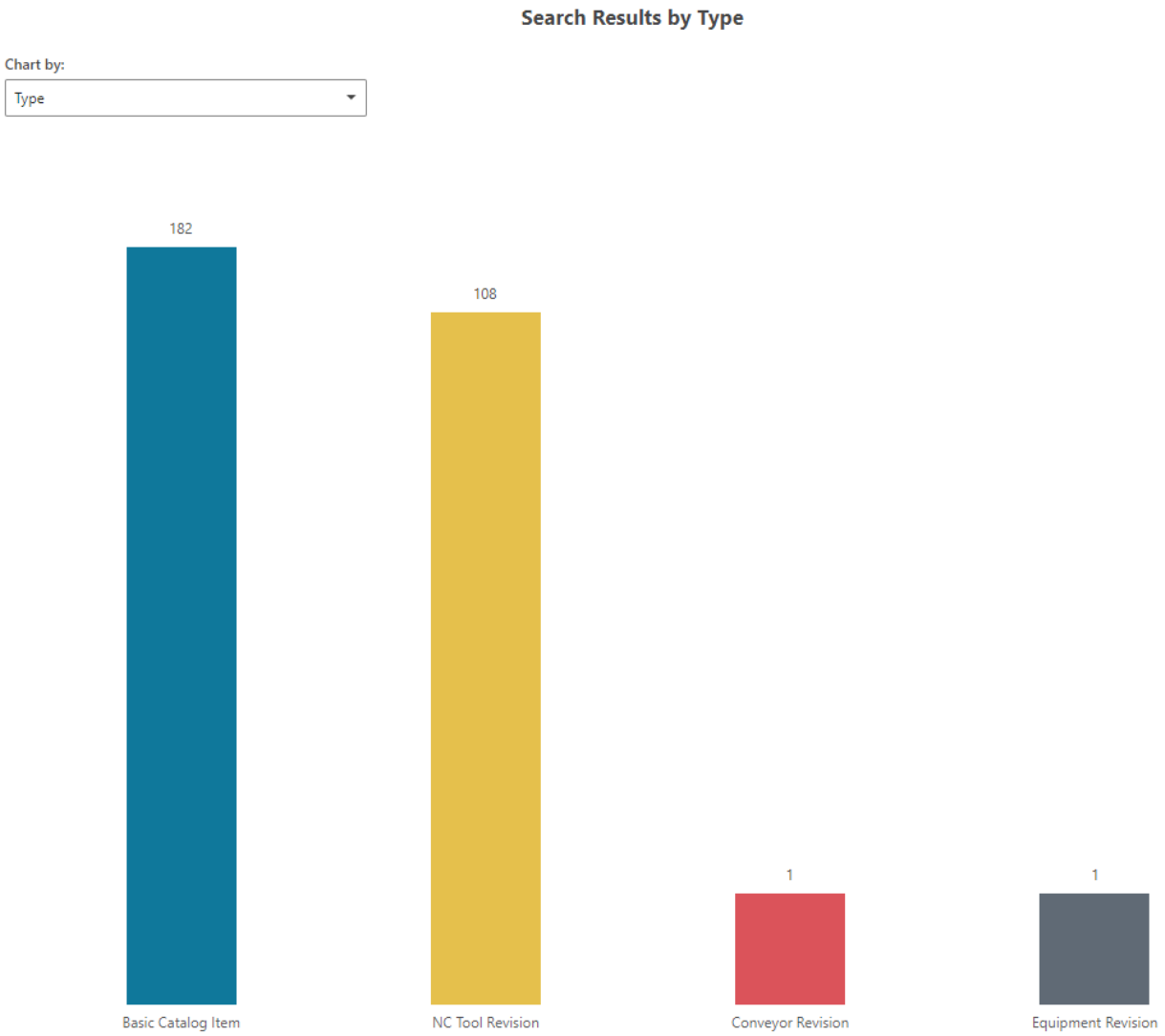

Use filters to limit the number of resources shown.

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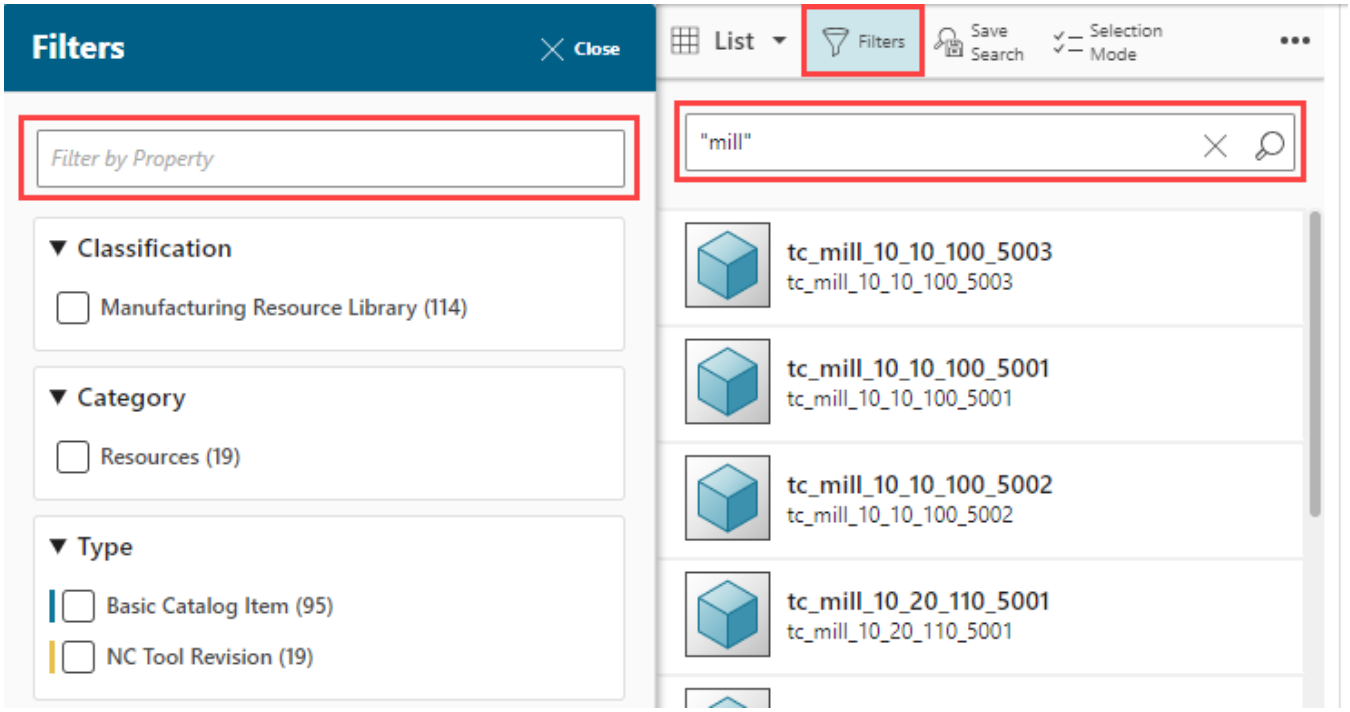
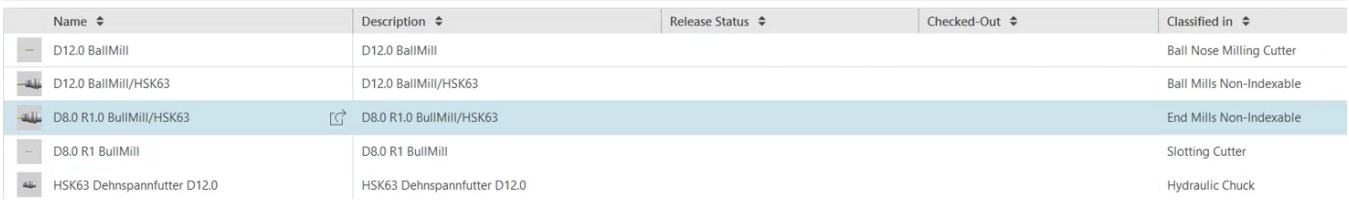


The MRL dashboard includes resources that you can view as a list, as a table, or as images.

> Note:
>
> When you display the table view, you can sort the resource and attribute columns by clicking a column header.

You can open a resource by clicking the **Open** icon next to the resource name.



Name	Description	Release Status	Checked-Out	Classified in
D12.0 BallMill	D12.0 BallMill			Ball Nose Milling Cutter
D12.0 BallMill/HSK63	D12.0 BallMill/HSK63			Ball Mills Non-Indexable
D8.0 R1.0 BullMill/HSK63	D8.0 R1.0 BullMill/HSK63			End Mills Non-Indexable
D8.0 R1 BullMill	D8.0 R1 BullMill			Slotting Cutter
HSK63 Dehnspannfutter D12.0	HSK63 Dehnspannfutter D12.0			Hydraulic Chuck

These attribute columns appear in the dashboard by default:

- **Name**
- **Description**
- **Release Status**
- **Checked-Out**

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- **ID**
- **Revision**
- **In Process**
- **Classified in**
- **Checked-Out By**

These attribute columns do not appear in the dashboard by default:

- **Alias ID**
- **Alternate ID**
- **Smart Discovery Indexed**

> Note:
>
> You can sort the resource columns by these attributes. You can also click the **Settings** icon, and then select **Arrange** to choose the columns to display.

View Manufacturing Resource Library data in Active Workspace

Additional steps may need to be performed by an administrator if the Manufacturing Resource Library (MRL) classes and objects are not visible when you change from using in Teamcenter in the Rich Client to using MRL in Active Workspace. For more information, see Deploying and administering basic classification.

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3. Finding and visualizing resources

Open existing resources

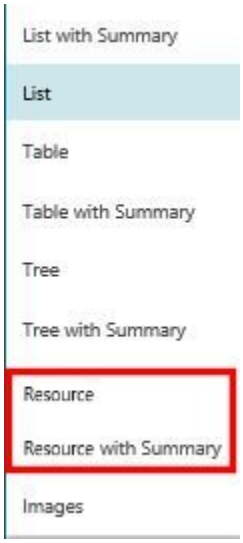
Resource Manager provides you with two ways to open existing resources. You can:

- Use the quick search at the top of the Active Workspace interface.
- Click the Classification tile to access the **Class Navigator**, where you can search for an existing component. Drill down into the classification structure or use the **Filter** input field.

Structure display options for item type Resource

You can easily examine resource structures in Teamcenter Active Workspace, including the way components are assembled from the machine side to the workpiece side, by using the **Resource** or **Resource with Summary** options.

The **Resource** and **Resource with Summary** options are available for item type **Resource** and for **Resource** subtypes, such as **NC Tool**.



Once you select **Resource** or **Resource with Summary**, the appropriate icon is displayed at the top of the diagram.

View mode icons:

Resource Mode	
Resource Summary Mode	

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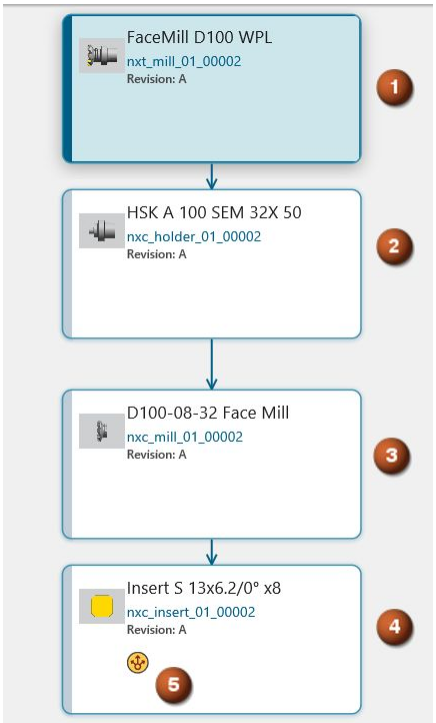
The **Resource** mode shows the resource structure as it is mounted from the machine side to the workpiece side.



1. Resource assembly
2. Tool holder
3. Cutter
4. Inserts
5. Information contained in each node:
    - Preview image (depending on availability, the image will be one of these items; preview item revision image, class icon, symbol of item type)
    - Item name
    - Item ID
    - Revision ID
    - Propagation start point (if component has a PSP assigned)

The **Resource with Summary** mode shows the resource structure diagram and in the secondary work area on the right, summary information.

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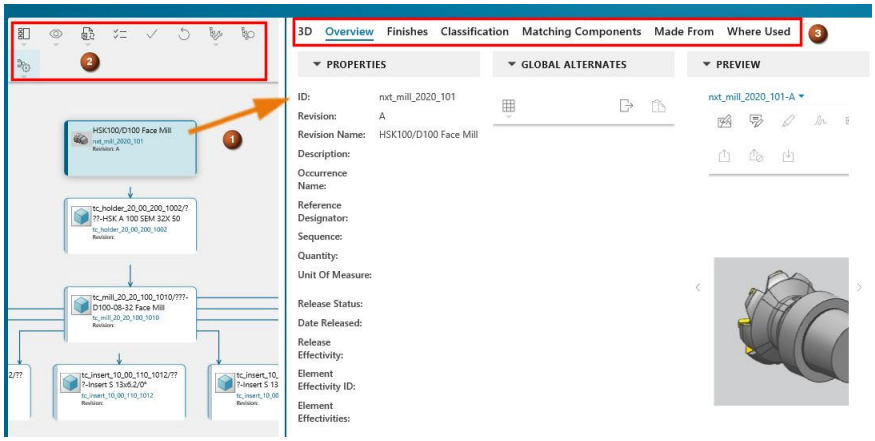


1. The information for a node that you select in the **Resource** is displayed in the **Overview** tab in the secondary work area.

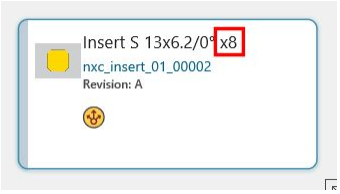
2. Commands that control the display of the **Resource Item Type** are shown above the **Resource** window.

3. Tabs that contain information about the selected node are shown at the top of the secondary work area.

Pack and unpack components

Some resources use multiple occurrences of the same component in the assembly structure. A typical example is when a resource assembly uses several cutting inserts of the same type.

The system displays these occurrences in a single node by default. When packed, a single node is displayed and the number of occurrences is shown in the box.



> Note:
>
> The component that occurs multiple times must have the same **Item ID** and **sequence number** in each instance for it to pack successfully.

Selecting **Unpack** separates each occurrence of the component into a separate node.

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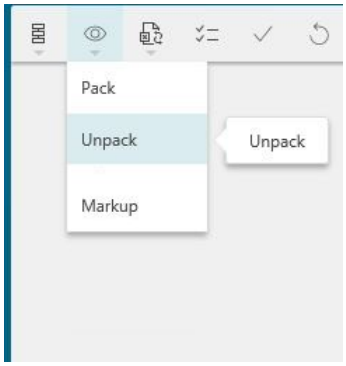
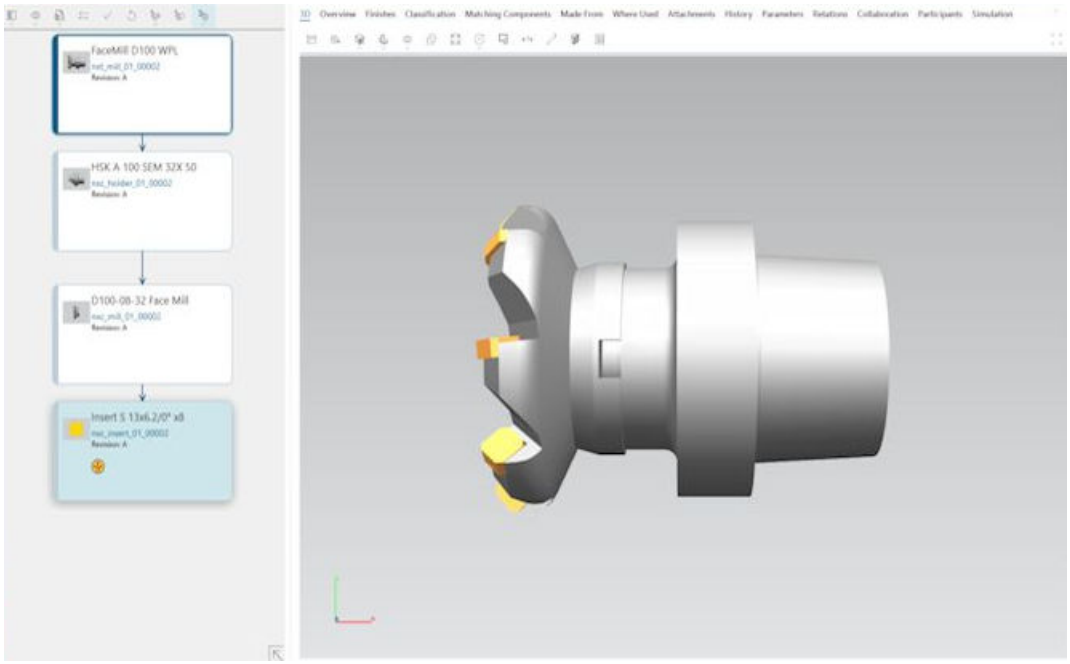


Resource with Summary synchronized with 3D Viewer

When you view the **Resource with Summary**, you can select the **3D** tab to display the 3D **Viewer**. If you select a component node in the diagram, the component is highlighted in the 3D **Viewer**. If you select a component in the 3D **Viewer**, the component node is highlighted in the diagram.



Note:

To add any custom item type to the **Content** tab to show all components of an assembly structure, follow this path: **Deployment and Configuration > Configuration and Extensibility > Configuring Active Workspace features > Active Content configuration** in the **Active Workspace Help** and see the Help topic **Marking archetypes to support structure**.

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Search components

Saving and recalling saved searches

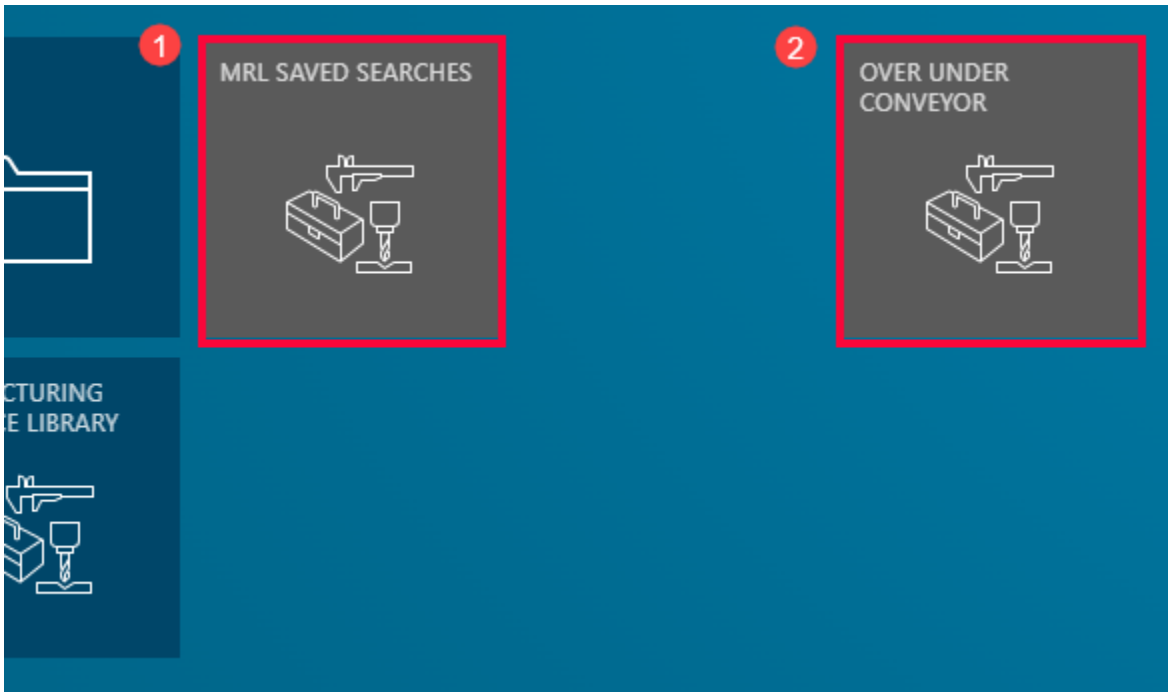
Saving a search

Searches in the Manufacturing Resource Library (MRL) can be saved to Active Workspace for quick access. You select **Pin to Home** in the **Save Search** panel.

Recalling a saved search

You can recall a saved search in one of the following ways:

- Access saved searches from the **Home** page:



1. Access saved searches from the **MRL Saved Searches** tile.
2. Access saved searches from the tile of a saved search that was pinned to the **Home** page.

- Access saved searches from the MRL dashboard:

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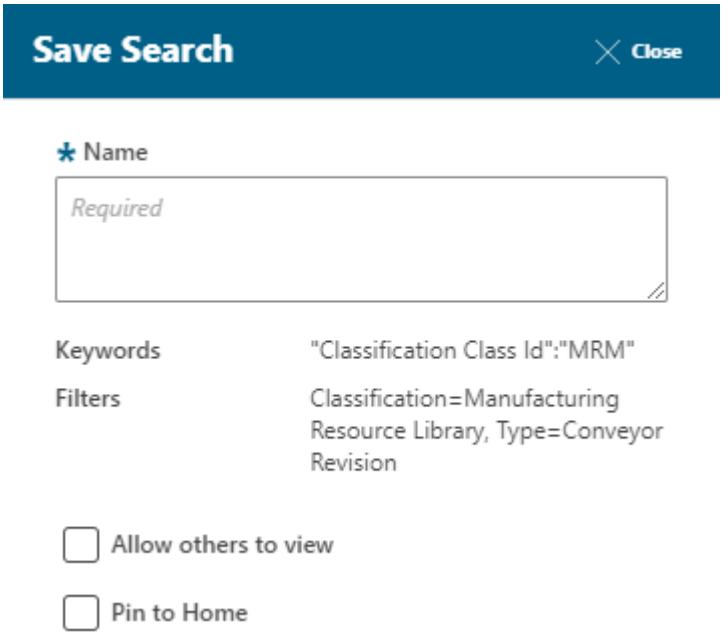
3 Access saved searches from **Saved Searches** on the MRL dashboard header.

Save a search

You can save searches to allow others to view or to quickly recall later.

1. Click the Manufacturing Resource Library tile on Active Workspace.
2. Select **Filters** at the top of the results panel to narrow the displayed resources to those you want to include in your search.
3. Click **Save Search** at the top of the results panel.
4. Enter the following information in the **Save Search** dialog box.



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Name	Type the name of the new resource. It is recommended that you make the name unique so users can easily search the database using this term.
Allow others to view	(Optional) To allow others to view your saved search.
Pin to Home	(Optional) To create a tile of the saved search on Active Workspace for quick access.

5. Click **Save**.

   A message confirms the search is saved.

Visualize resources in 3D in Resource Manager

Visualizing resources in 3D viewer

You can use the **3D** viewer to manipulate views, to query and measure parts and features, to view part and assembly level product and manufacturing information (PMI), and to create 3D sections through your resources.

For more detailed information about the 3D Visualization capabilities in Active Workspace, see **Visualizing 3D Product Data** on the Siemens Support Center.

You can explore 3D data (JT) associated with resources in Resource Manager under either the **Part Manufacturing** tab or the **Content** tab.

3D tab under the Part Manufacturing tab

The **3D** tab is available for individual items in a resource assembly in the **Part Manufacturing** tab.

> Note:
>
> The available tabs and their order can differ depending on what templates were imported and where you are in Active Workspace.

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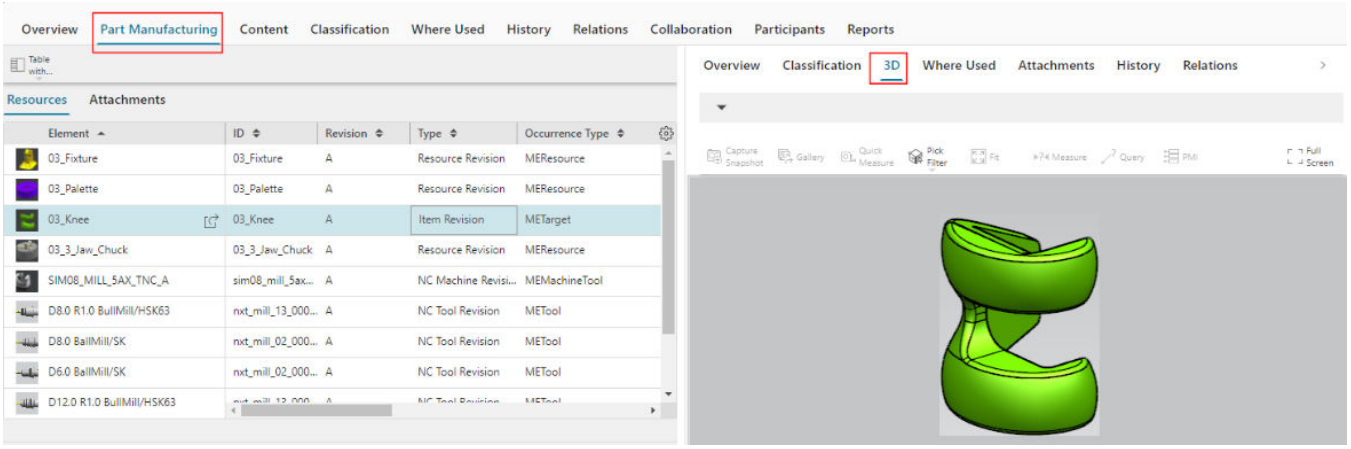
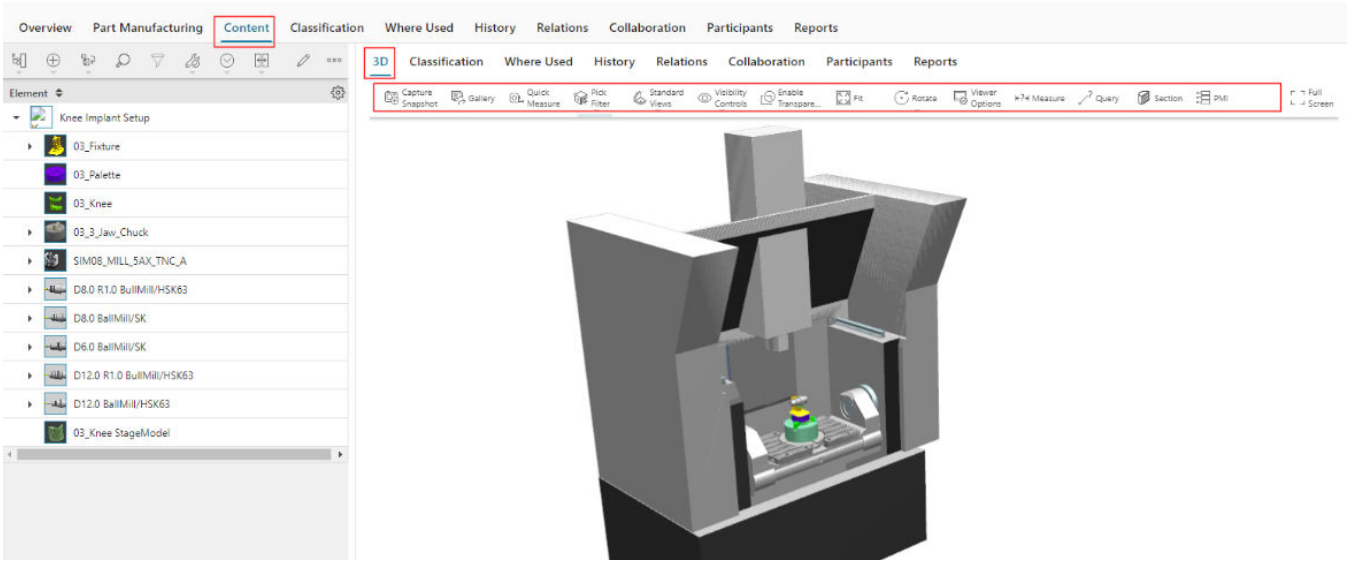


3D tab under the Content Tab

The **3D** tab is available for the full setup in the **Content** tab for the resource type **NC Machining Revision**.



Under the **Content** tab, you can highlight an individual component in the full setup by selecting the component in the structure on the left or the graphic on the right.

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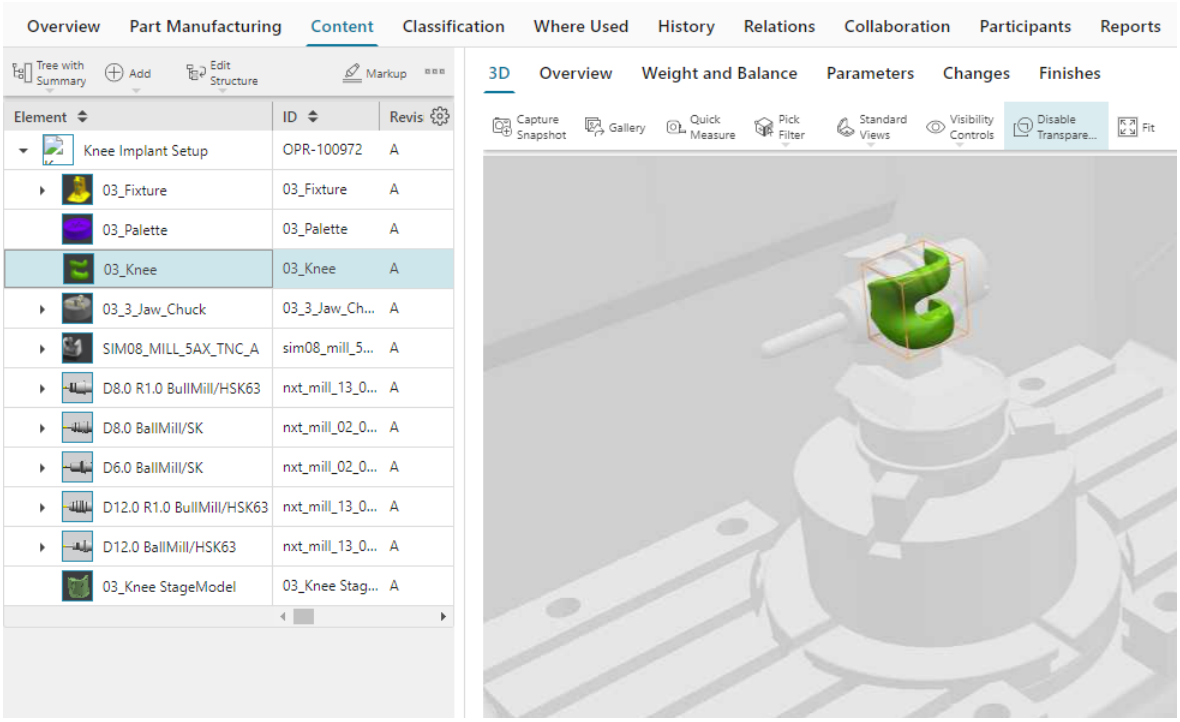

Features for controlling 3D display

	Gallery	Show a collection of snapshots in the 3D view.
	Quick Measure	Calculate temporary measurements between the parts or features in an assembly. The default unit is meters.
	Pick Parts/ Pick Features+	Calculate distances between the selected parts or selected features such as edges, points, or arc centers.
	Standard Views	Align the 3D view of the model to a standard angle. You can choose from the following views: **Top**, **Bottom**, **Left**, **Right**, **Front**, **Back**, **+Isometric**, **-Isometric**.
	Visibility Controls	Show or hide parts and subassemblies. **All On**, **All Off**, **Remove Analysis Results**
	Enable Transparent Selection Display Mode	**Enable** to view the selected parts in their original color and with bounding boxes. Unselected parts appear transparent. **Disable** to view the selected parts in the highlight color and the unselected parts in their original color.
	Fit	Fit all of the components to the work area. Selecting specific components fits the selection to the work area.

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	Pan, Rotate, Zoom	**Pan**—Move the 3D view vertically and horizontally with the mouse to change the center of the view. **Rotate**—Rotate the view of the 3D model around the center of the view with the mouse. This is the default navigation mode. **Zoom**—Zoom in or zoom out of the current work area with the mouse.
	Viewer Options	Control the view's characteristics, such as the material, units, and 3D navigation model.
	Measure	Calculate the minimum distance between the selected parts of features.
	Query	Display geometric information of the selected parts or features.
	Section	Create and manage sections used to clip portions of the 3D geometry in the 3D view. This allows users to view internal elements to expose conditions such as interference and clearance issues.
	PMI	Show and hide specific PMI or apply Model Views.
	Full Screen	Switch to the full-screen view.

Visualize individual resources in 3D viewer

1. Open a resource with the resource type **NC Machining Revision**.
2. Select the **Part Manufacturing** tab.
3. Select an item from the list of resources in the **Part Manufacturing** tab, and then select the **3D** tab.

> Note:
>
> The **3D** tab is only available for resources that have 3D data.

4. Adjust the display of the structure using the features at the top of the graphics window in the **3D** tab.

Visualize resource setup in 3D

1. Open a tool resource.
2. Select the **Content** tab.

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3. Select the resource assembly line and then select the **3D** tab.

4. Adjust the display of the structure using the features at the top of the graphics window in the **3D** tab.

5. (Optional) Select an individual resource in the assembly to highlight it in the graphics view.

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4. Populating the manufacturing resource library

Creating resources

Creating a resource

You can create new or revised resources in the Manufacturing Resource Library (MRL) workspace in the following ways:

- Create a new resource without classification or attributes.

  When a resource is not selected or opened, use **More commands** •••→**New** ✳→**Create Resource** ⊕ on the primary toolbar to create and manually classify a new resource.

  For more information see **Create new resources**.

- Create a new resource and select to use the classification and attributes of an existing resource.

  When a resource is selected or opened, use **More commands** •••→**New** ✳→**Create Resource** ⊕ on the primary toolbar to create a new resource, but you can select to use the classification and values of the selected resource.

  For more information see **Create resources using an existing resource**.

- Create a new resource that includes the classification and attributes of an existing resource.

  When a resource is selected or opened, use **More commands** •••→**New** ✳→**Save As** on the primary toolbar to create a new resource with the classification and values of the selected resource.

  For more information see **Create resources using Save As**.

- Create a revision of an existing resource that includes the classification and attributes.

  When a resource is selected or opened, use **More commands** •••→**New** ✳→**Revise** on the primary toolbar to revise a resource with the classification and values of the selected resource.

  For more information see **Create resources using Revise**.

- Create a revision of an existing resource and all parent assemblies that includes the classification and attributes.

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When a resource is selected or opened, use **More commands** •••→**New** ✳→**Revise Including Parent Assemblies** on the primary toolbar to revise a resource, and all parent assemblies, with the classification and values of the selected resource.

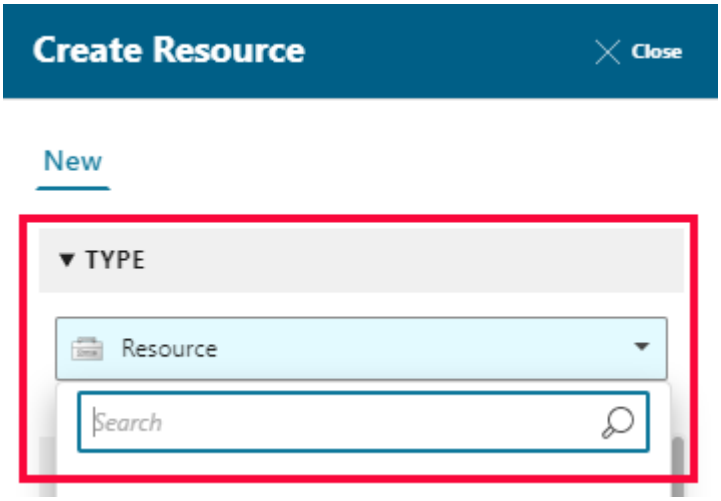
For more information see **Revise resources and parent assemblies**.

> Note:
>
> New and revised resources can be accessed in the **New Resources** tab on the Manufacturing Resource Library (MRL) dashboard.

Create new resources

Create new resources without classification or attributes in the Manufacturing Resource Library (MRL) dashboard workspace.

1. On Active Workspace, click the **Manufacturing Resource Library** tile.
2. Select **More commands** •••→**New** ✳→**Create Resource** on the primary toolbar.
3. Select the resource type from the **TYPE** list, or search for it in the **Search** field.



4. Enter the **Properties** for the new resource.

ID	Manufacturing Resource Library (MRL) automatically assigns a unique ID that you can accept or change. If you change it, the new ID must be unique.
Revision	MRL automatically assigns a revision letter that you can accept or change.

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Name	Type the name of the new resource. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the new resource.

5. Click **Create**.

Note:

- Once the resource is created, it is recommended that you next classify the resource.
- If this resource is a root node for a resource assembly, you can now add components.

Create resources using an existing resource

Create new resources with the classification or values of the selected resource in the Manufacturing Resource Library (MRL) dashboard workspace.

1. On Active Workspace, click the **Manufacturing Resource Library** tile.
2. On the MRL dashboard workspace, select or open a resource that includes the classification or values to be used for the new resource.
3. Select **More commands** •••→**New** ✳→**Create Resource** ⊕ on the primary toolbar.
4. In the **Create Resource** dialog box, you can select the following:
   - Select (1) **Classify in Same Class** to classify the new resource in the same class as the selected resource.
   - Select (2) **Copy Values** to copy the values of the selected resource to the new resource.

   Note:

   Copy Values can be selected only when **Classify in Same Class** is selected.

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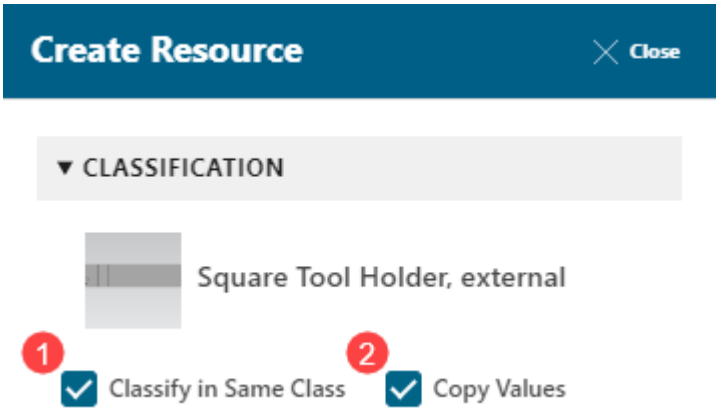
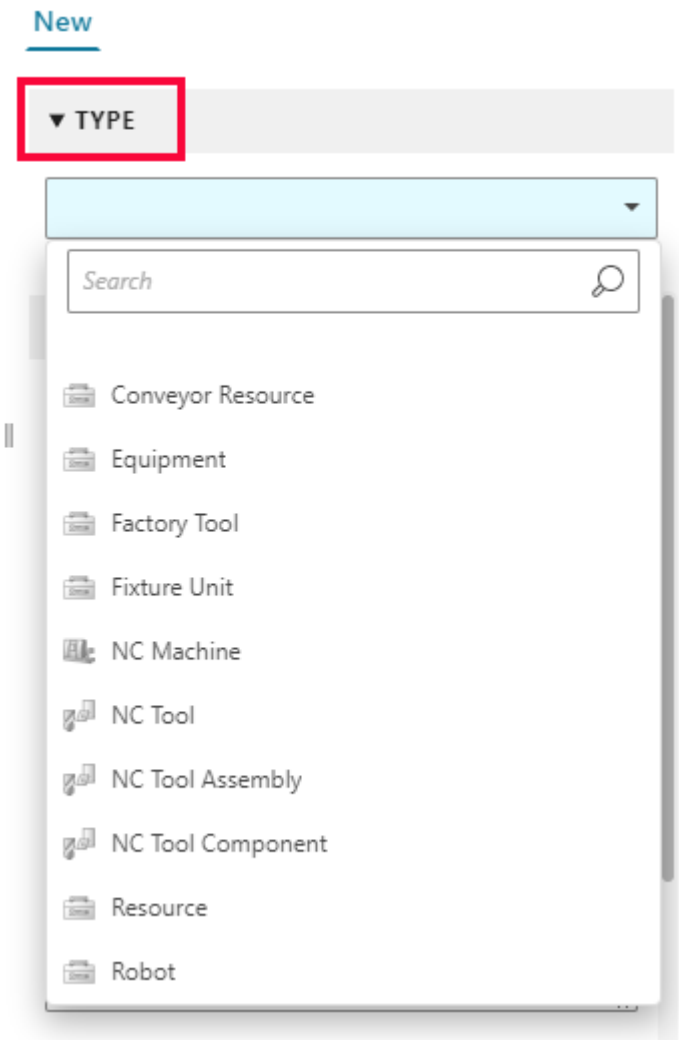


5. Select the resource type from the **TYPE** list, or search for it in the **Search** field.



6. Enter the **Properties** for the new resource.

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ID	MRL automatically assigns a unique ID that you can accept or change. If you change it, the new ID must be unique.
Revision	If this is a revision of an existing resource, type the next consecutive revision letter.
Name	Type the name of the new resource. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the new resource.

7. Click **Create**.

> Note:
>
> If the existing resource is an assembly, the new resource will inherit the components, classification object (ICO), and structure of the existing resource.

Create resources using Save As

Create new resources with the classification or attributes of the selected or opened resource in the Manufacturing Resource Library (MRL) workspace.

Procedure

1. On Active Workspace, click the **Manufacturing Resource Library** tile.

2. On the MRL workspace, select or open a resource that includes the classification or values to be used for the new resource.

3. Select **More commands** •••→**New** →**Save As** on the primary toolbar.

4. In the **Save As** dialog box, enter the **Properties** for the new resource.

Name	(Optional) Type the name of the new resource. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the new resource.

5. Click **Save**.

> Note:
>
> If the existing resource is an assembly, the new resource will inherit the components, classification object (ICO), and structure of the existing resource.

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Create resources using Revise

Create revised resources with the classification or attributes of the selected or opened resource in the Manufacturing Resource Library (MRL) workspace.

Procedure

1. On Active Workspace, click the **Manufacturing Resource Library** tile.
2. On the MRL workspace, select or open a resource that includes the classification or values to be used for the revised resource.
3. Select **More commands** •••→**New** ✳→**Revise** ⟲ on the primary toolbar.
4. In the **Revise** dialog box, enter the **Properties** for the revised resource.

Revision	MRL automatically assigns the next consecutive revision letter that you can accept or change.
Name	(Optional) Type the name of the new resource. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the revised resource.

5. Click **Revise**.

> Note:
>
> If the existing resource is an assembly, the revised resource will inherit the components, classification object (ICO), and structure of the existing resource.

Adding vendor catalogs

Working with a vendor catalog

So you can use tools from a vendor catalog, you must download a file or data from the tool vendor and import it into the Manufacturing Resource Library (MRL).

You can select (1) tool vendor product data, 3D data (STEP files), and tool components, such as drills, holders, inserts, and adapters, from various tool vendors, using the (2) **Generic Tool Catalog (GTC)** or **DIN** format, and download them to (3) your system.

The tool vendor product data or catalog package files can then be (4) imported directly into the MRL.

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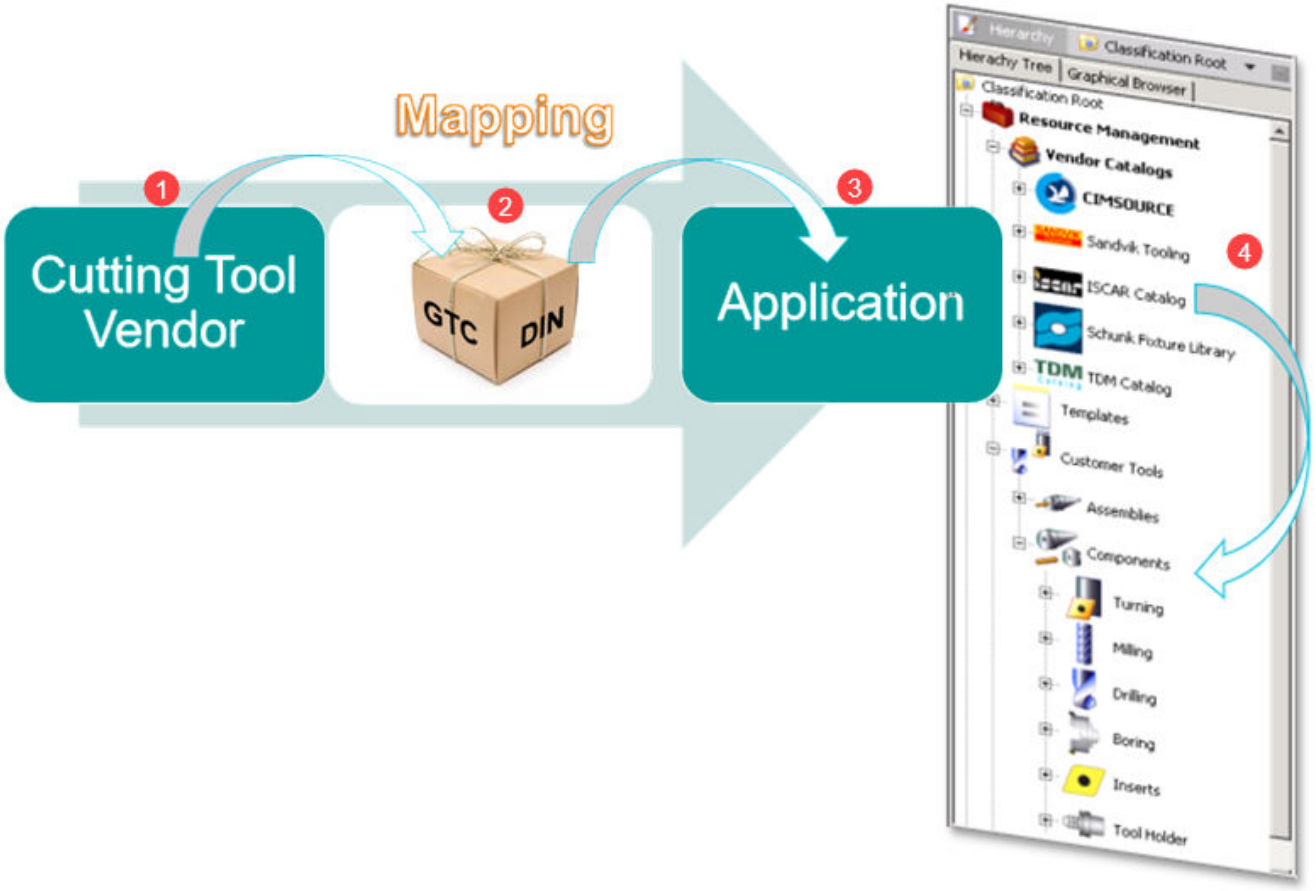

When you import a tool vendor catalog package to the MRL, the system automatically assigns a new object identifier for the mapped object.

> Note:
>
> When you download a **DIN** tool vendor catalog package from a tool vendor, it is possible to enter a customer-specific product identifier that you can assign to the mapped object, instead of the system generated identifier.
>
> This feature can be activated or deactivated. It is activated by default.

- For **GTC** vendor catalog packages, a **GTC** vendor catalog package hierarchy must first be uploaded before the vendor product data can be imported. For more information see **Import GTC vendor class hierarchy**.

- For **DIN** vendor catalog packages, the option to populate the vendor catalog package hierarchy must be selected in the Teamcenter Configuration Setup before the vendor product data can be imported.

  If installing to the MRL for the first time, see Import the class hierarchy.

  If upgrading from an earlier MRL version, see Update the class hierarchy.

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Note:

For more information on installing or updating the Manufacturing Resource Library, see Installation overview and workflow.

The **DIN** formats supported for tool vendor catalog packages are **DIN 26100:2017-11** and **DIN 26100:2021-05**.

To upload larger vendor catalog files, it is best to adjust the Active Workspace upload timeout setting in file path: *AW ROOT/microservices/gateway-nnn/config.json*.

Note:

After making changes to the *AW ROOT/microservices/gateway-nnn/config.json* file, you must restart the gateway.

- The default upload timeout setting, in milliseconds, is **httpRequestResponse: 1800000**. The recommended setting is **httpRequestResponse: 7200000**.

```
"timeout": { "httpRequestResponse": 600000, "autoLogout": "30s",
   "proxyRequest": "5s" },
```

It is also recommended that you set **QUERY_TIMEOUT** to **0** in the *serverPool.properties* file so that the server connection does not time out automatically.

After making changes to the *serverPool.properties* file, you must restart service *Teamcenter Server Manager config1_PoolA* to implement the changes.

The tool components can be automatically mapped to existing MRL tool classes.

You can build a tool assembly based on these components, create graphics for it, send it to NX, and use it to machine parts in NX.

Import GTC vendor class hierarchy

Import a GTC vendor product data hierarchy into the Manufacturing Resource Library (MRL).

Note:

Before a user can import **GTC** vendor product data, an administrator or dba user must first import a vendor class hierarchy to the MRL. For additional information see **Working with a vendor catalog**.

1. On Active Workspace, click one of the following tiles:

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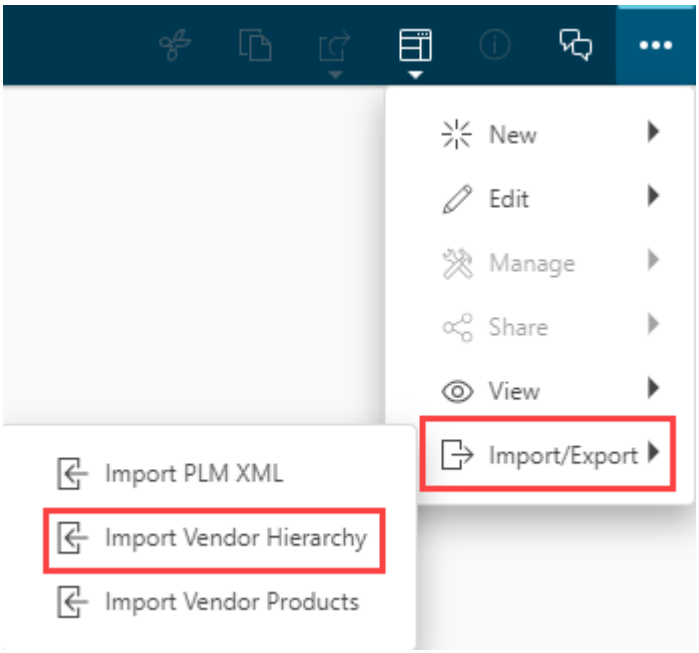
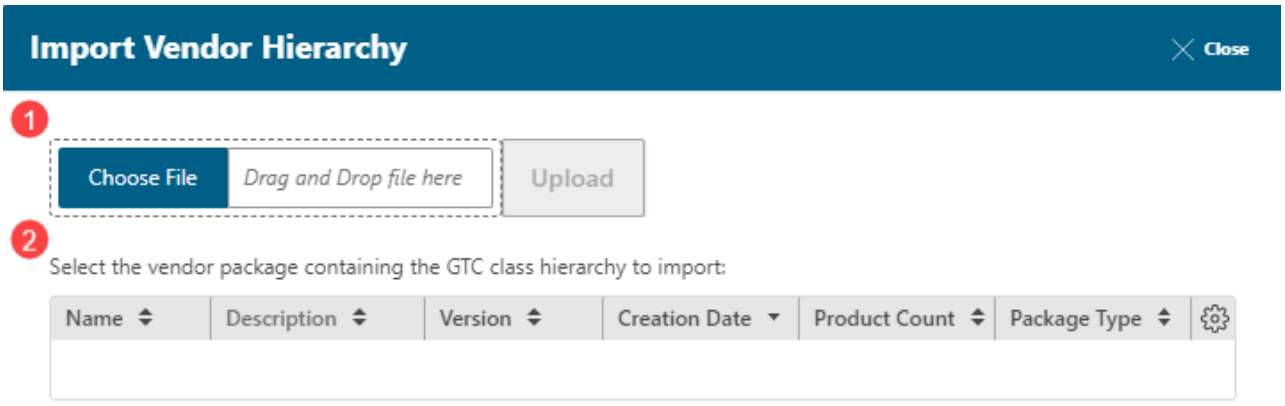
- **Manufacturing Resource Library**
- **Classification**
- **Explorer**

2. On the primary toolbar, click the **Import/Export** icon and select **Import Vendor Hierarchy**.



3. In the **Import Vendor Hierarchy** dialog box, do one of the following to select the product hierarchy:



- If needed, select (1) **Choose File** to search for the vendor hierarchy package ZIP file that contains the product hierarchy data, and click **Upload**.

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> Note:
>
> Depending on the size of the ZIP package you select, this action can take time to upload.

- Select the (2) vendor tool catalog hierarchy you want to import from the list of available packages.

4. Click **Import Hierarchy**.

5. (Optional) Open the resulting log file to view the import state of individual components.

Import vendor product data

Import vendor product data and tool components into the Manufacturing Resource Library (MRL).

> Note:
>
> Before a user can import **GTC** or **DIN** vendor product data, an administrator or dba user must first import a vendor class hierarchy to the MRL. For additional information see **Import GTC vendor class hierarchy** or **Working with a vendor catalog**.

1. On Active Workspace, click one of the following tiles:

   - **Manufacturing Resource Library**
   - **Classification**
   - **Explorer**

2. On the primary toolbar, click the **Import/Export** icon and select **Import Vendor Products**.

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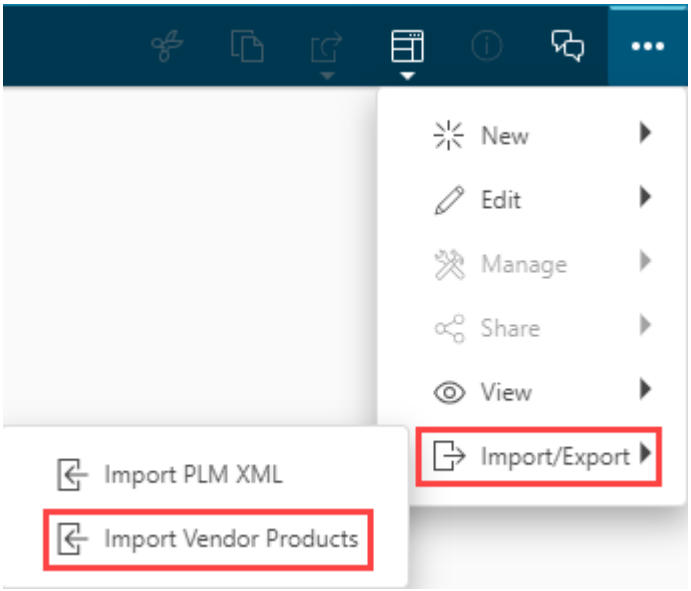
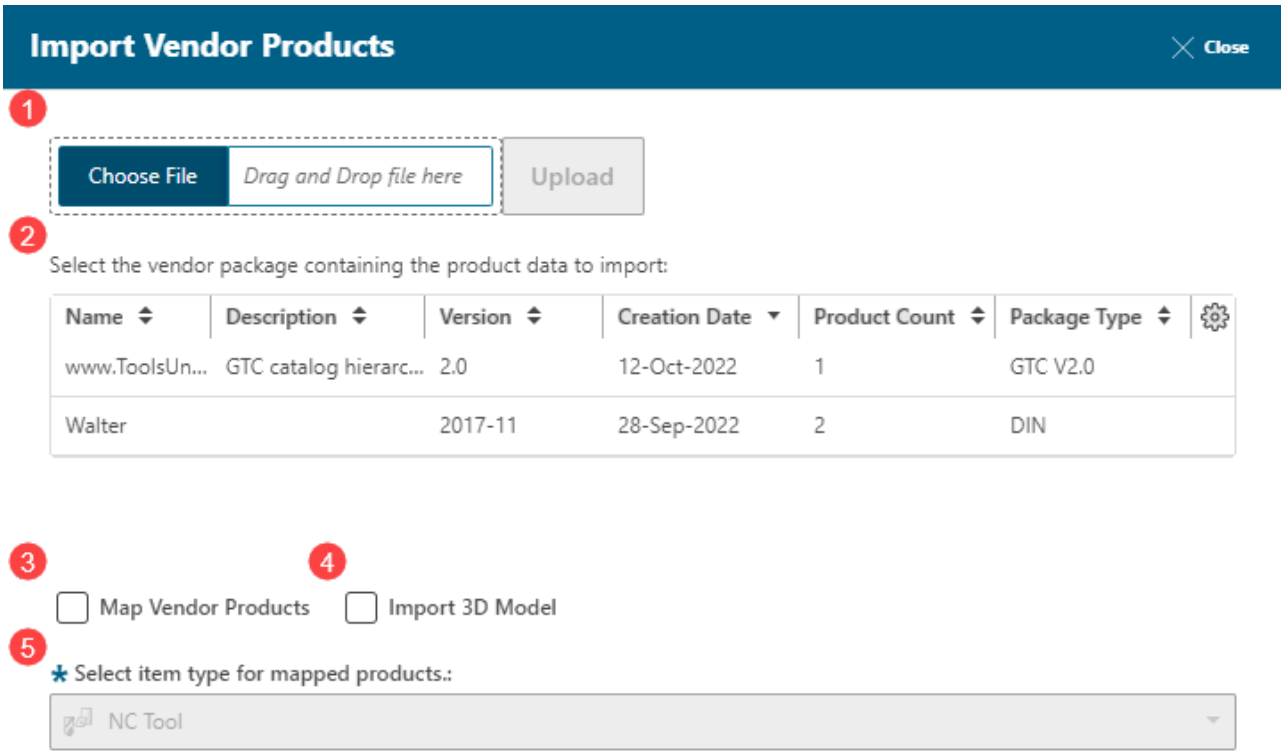


3. In the **Import Vendor Products** dialog box, take action as needed:



- If needed, select (1) **Choose File** to search for the vendor package ZIP file that contains the product data, and click **Upload**.

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> Note:
>
> Depending on the size of the ZIP package you select, this action can take time to upload.

- Select the (2) vendor tool catalog package you want to import from the list of available packages.
- Select the (3) **Map Vendor Products** check box to map the products from the vendor classes into the MRL tool component classes.
- Select the (4) **Import 3D Model** check box to import the 3D data (STEP files) in the vendor tool catalog package files, if available.

  Do not select the **Import 3D Model** check box when using the NX **Import Vendor 3D Graphics** feature in the **Manufacturing Resource Library** ribbon. For more information, see Managing tool assemblies between Manufacturing Resource Library and NX.
- Select the (5) item type in the **Select item type for the mapped products** list. You must first select **Map Vendor Products** to make the **Select item type for the mapped products** option available.

4. Click **Import Data**.

> Note:
>
> Depending on the size of the package you select, this action can take several hours to complete.

If multiple target classes are available for any of the imported items, the **Map Products for Multiple Target Classes** dialog box opens, allowing you to select the MRL tool component class.

5. In the **Map Products for Multiple Target Classes** dialog box, view the (1) **Source Class**, the tool component class assigned by the vendor, and then take action as needed:

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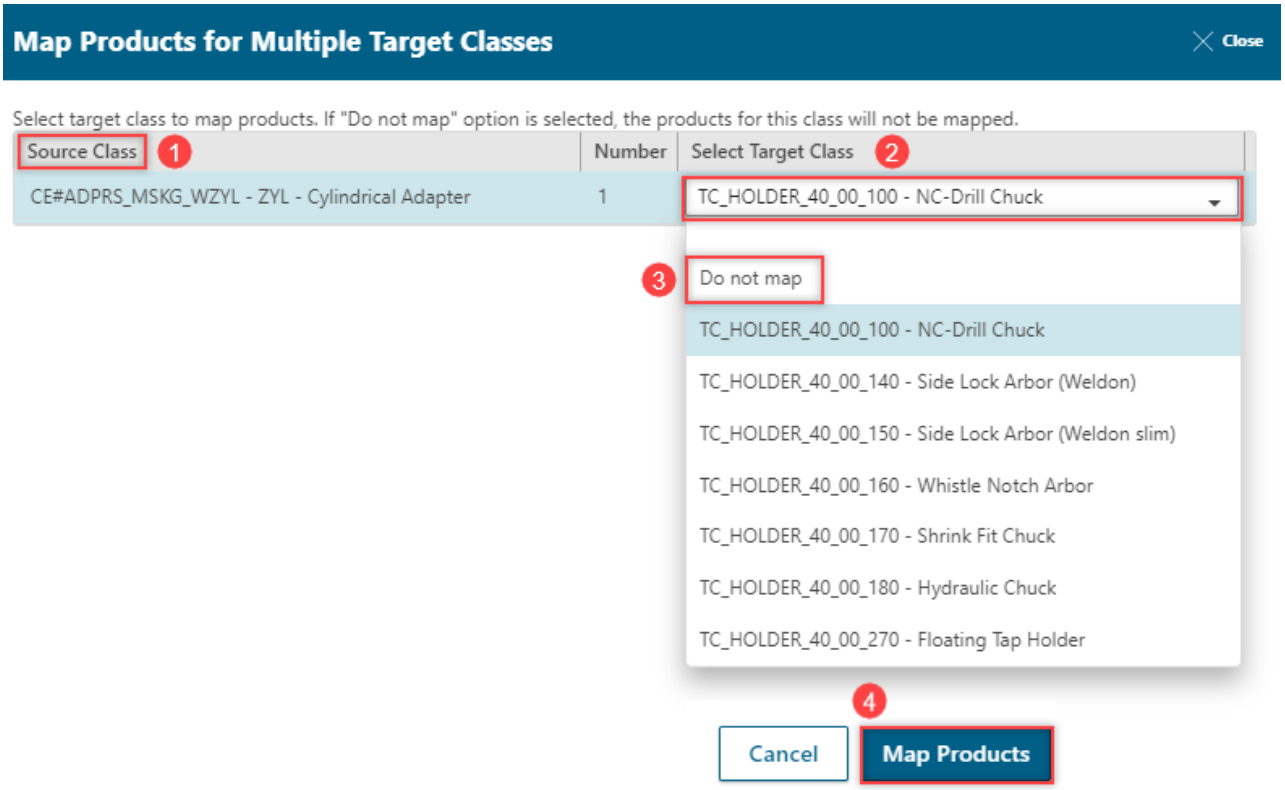

- In (2) **Select Target Class**, choose a different tool component class in MRL, for the selected product.
- Select (3) **Do not map** so that the tool component class for the component is not mapped in MRL.
- Select (4) **Map Products** to confirm the selected tool component classes in MRL.

6. Click **Map Products**.
7. (Optional) Click **Show Imported Products** to open the imported components in MRL.
8. (Optional) Click **Download Log File** to open the resulting log file, allowing you to view the import state of the individual components.
9. (Optional) Click **Show Mapped Products** to open the mapped components that were imported in MRL.

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Creating resource assemblies

Creating resource assemblies

In addition to individual components, Resource Manager manages the resource assemblies that are used by manufacturing tasks. Classified resource components are used to build these resource assemblies.

Generally, tool components cannot be used in a manufacturing process by themselves, so they are assembled using Resource Manager into resource assemblies. These assemblies are then used in the manufacturing production process. For example, parts are added to a resource assembly by a tool designer and then used in a manufacturing process created by a process planner.

There are three main features of a resource assembly:

- Assembly structure

  The assembly structure is displayed in a tree hierarchy on the left side of the Resource Manager application window in the resource view. The tree hierarchy shows the logical structure of how the different components are assembled to make up the structure. The root node represents the assembly. Additional components are added to the root node to build the structure.

  You can use cut, copy, paste, or drag-and-drop functions to organize components within the structure hierarchy.

- Assembly attributes

  If its root node is selected in the assembly structure, its attributes are shown on the **Classification Properties** view. If subassembly or component nodes are chosen, the attribute list in the **Classification Properties** view changes and the attributes of the selected object are displayed.

- Graphical representation

  The **Classification Properties** view has two viewing windows on its right-hand side. The upper window displays an image associated with its class, while the lower window shows an image associated to the instance, if available.

Create a tool resource assembly

Start with an existing resource or create a new one. In the steps below, you create the new root-node assembly item.

1. Click the **Folders** icon on the Active Workspace **Home** page.

2. Select **More commands** •••→**New** →**Add** ⊕ on the primary toolbar.

3. Search for **NC Tool** in the **Filter** field or select it in the **Recent** list if available.

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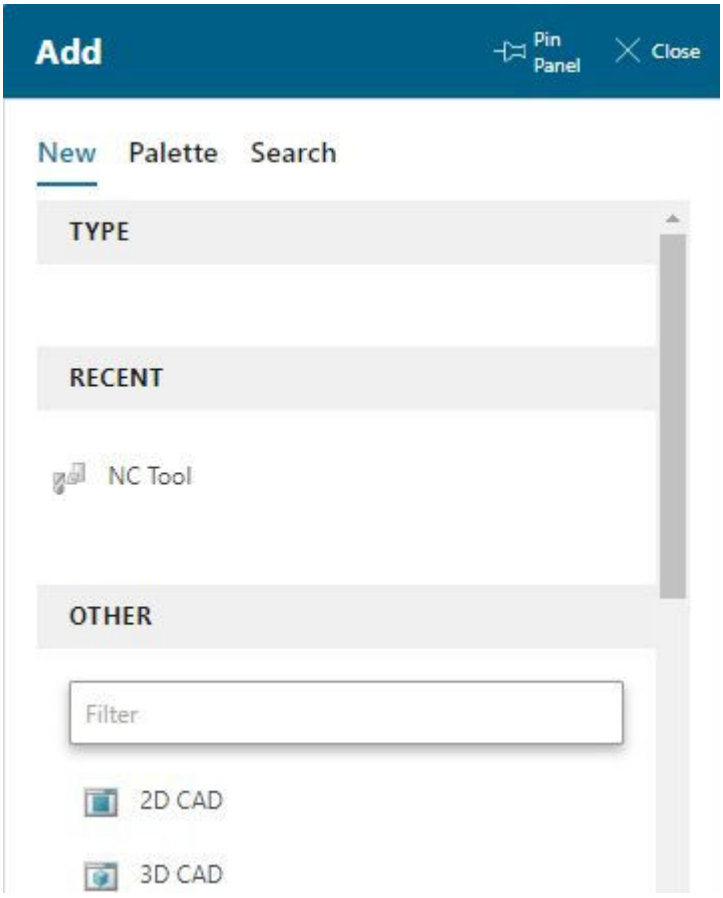

4. Enter a name for the new component in the **Add** dialog box.

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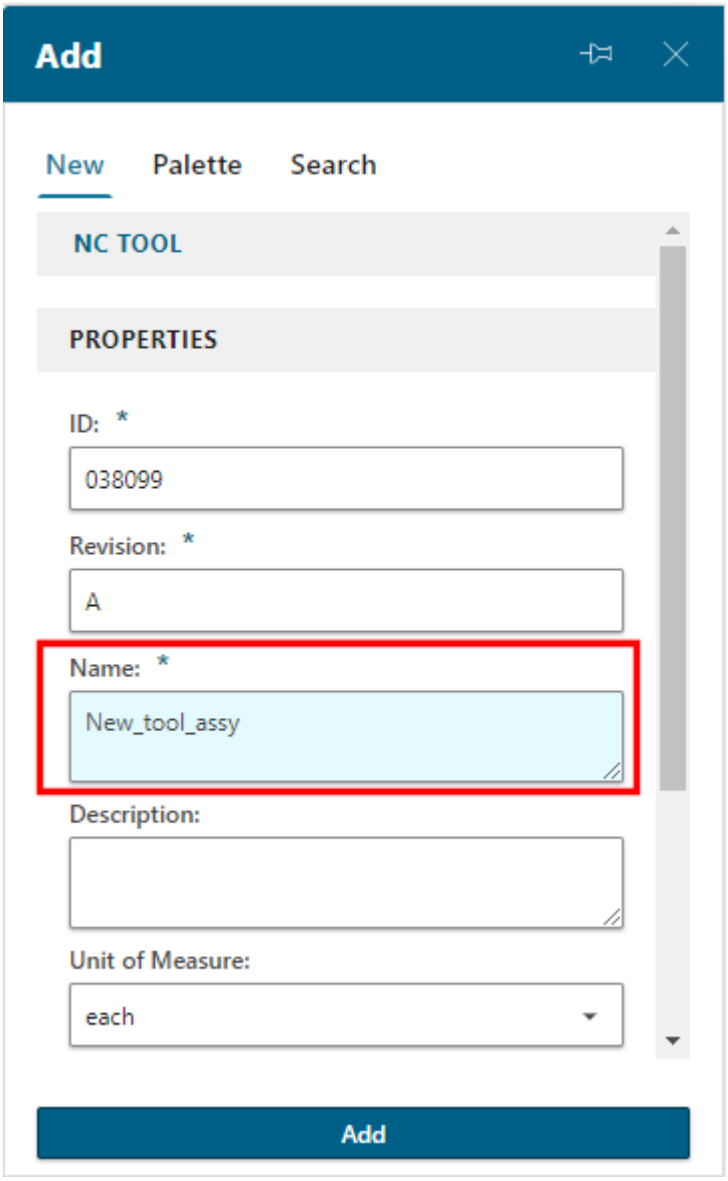

5. Click **Add**.

> Note:
>
> After you add a new component, you can classify it by selecting the **Classification** tile from the Active Workspace home page.

For more information on classifying resources, see Active Workspace, Advanced Classification — Deployment and Administration.

For more information on creating resources, see Create new resources.

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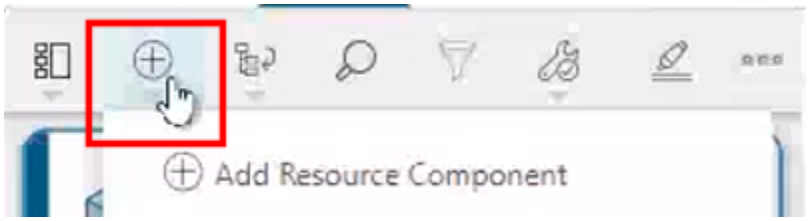
Methods for adding components to a tool resource assembly

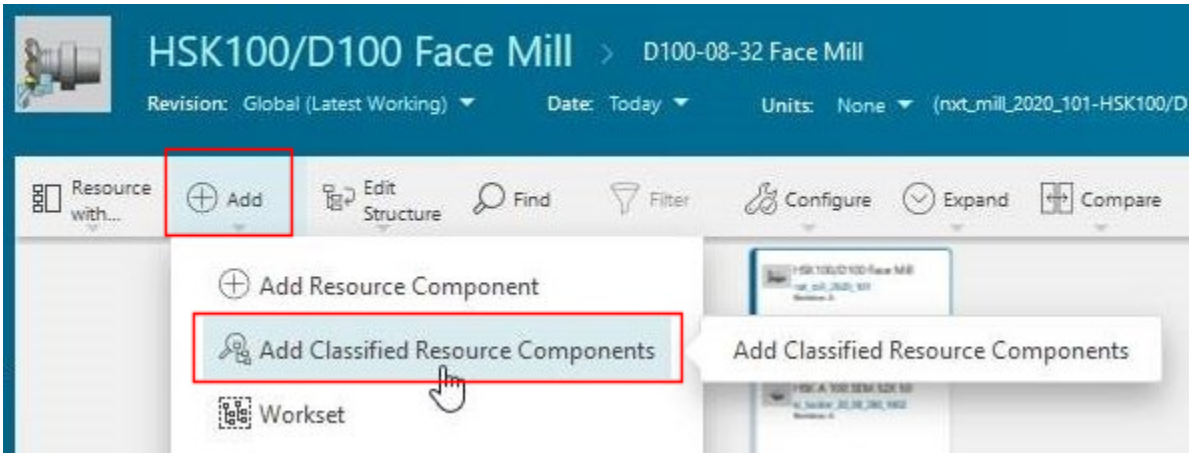
There are several ways to add a component to a resource assembly.

- You click the **Add Resource Component** ⊕ icon on the menu bar above the summary view.



From the **Add** menu, either add an existing resource or create a new resource.

- You select **Add Classified Resource Components**.



You then can search the **Classification Class Tree** hierarchy from the **Class Navigator**.

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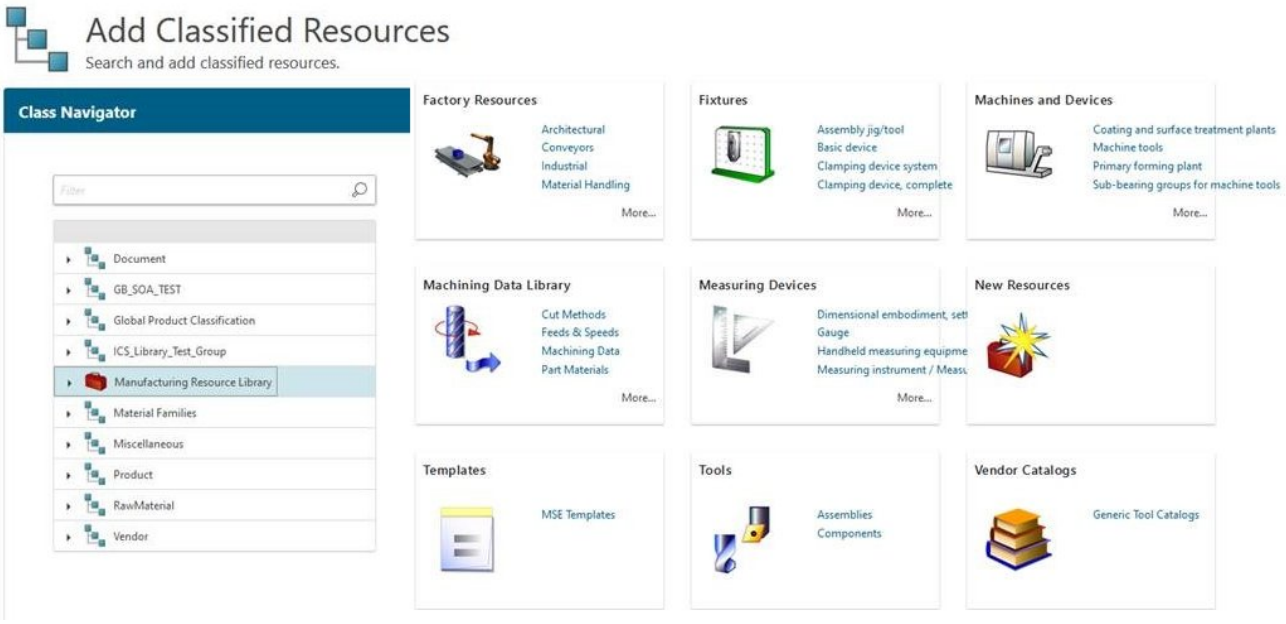
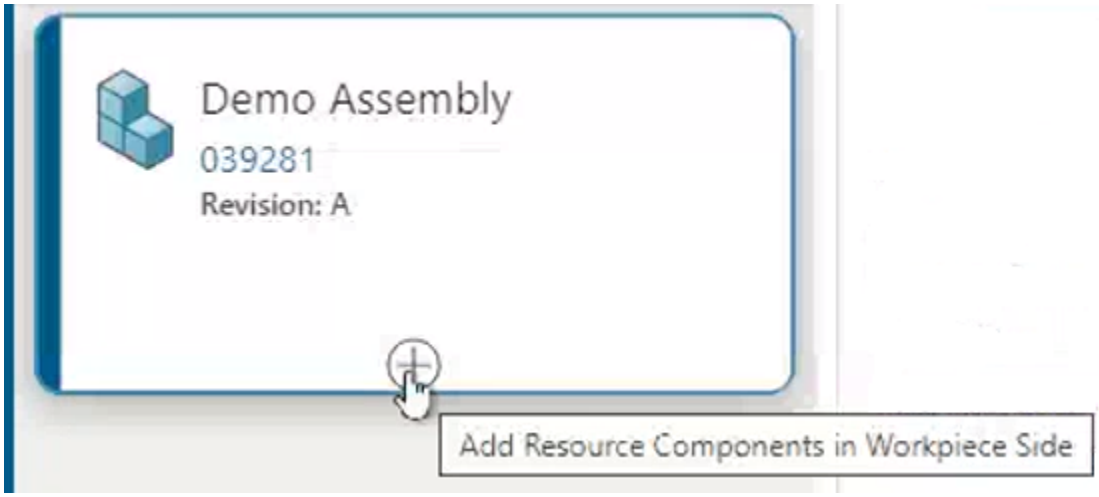


- You click **Add Resource Components in Workpiece Side** ⊕ from the assembly node.



From the **Add** menu, either add an existing resource or create a new resource.

- You add a matching component by selecting a component in the structure and selecting the **Matching Components** tab.

> Note:
>
> The available tabs and their order can differ depending on what templates were imported and where you are in Active Workspace.

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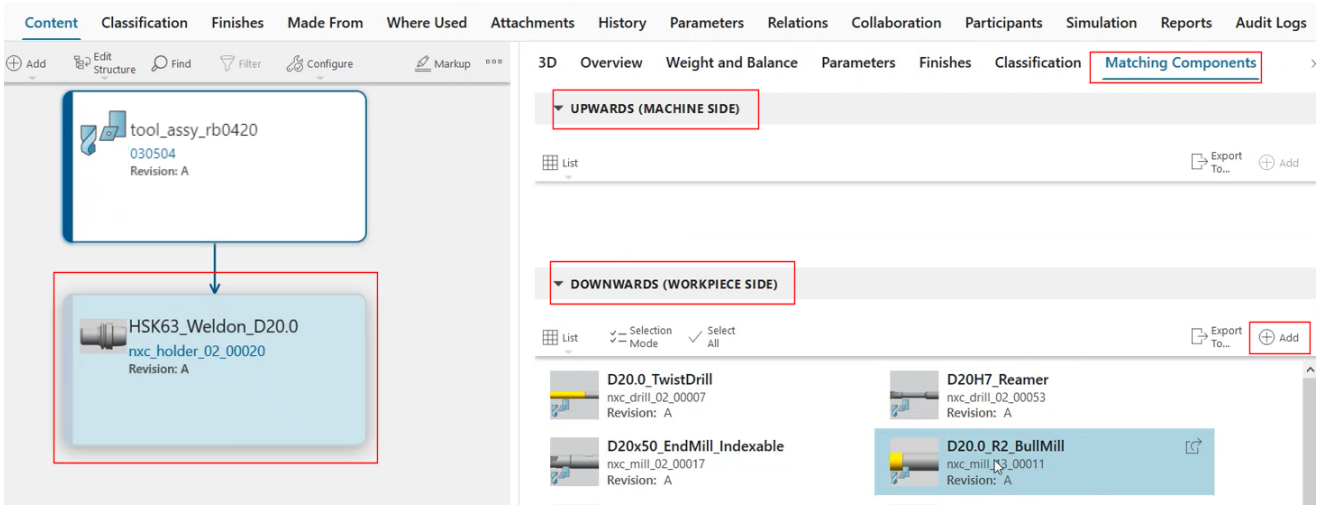
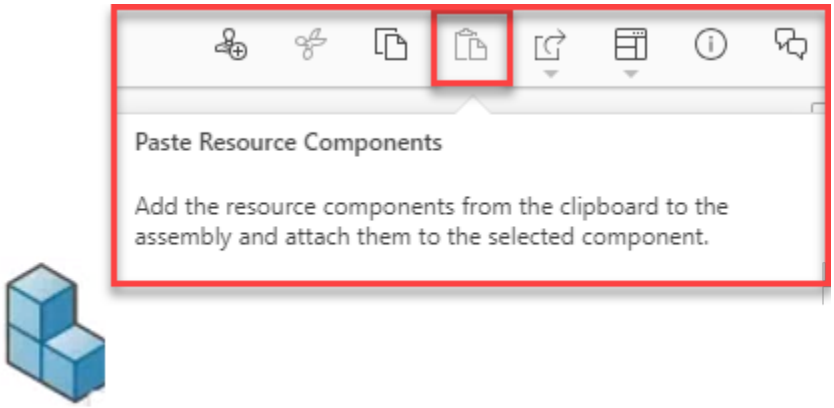


From the **Matching Components** tab, select a matching component from the matches in the machine side or workpiece side results, and click ⊕ **Add**.

- You can copy and paste a component.



Note:

- After you add a new component, you can classify it by selecting the **Classification** tile from the Active Workspace home page.

  For more information on classifying resources, see Active Workspace, Advanced Classification — Deployment and Administration.

- You must unpack a component before you add a matching component.

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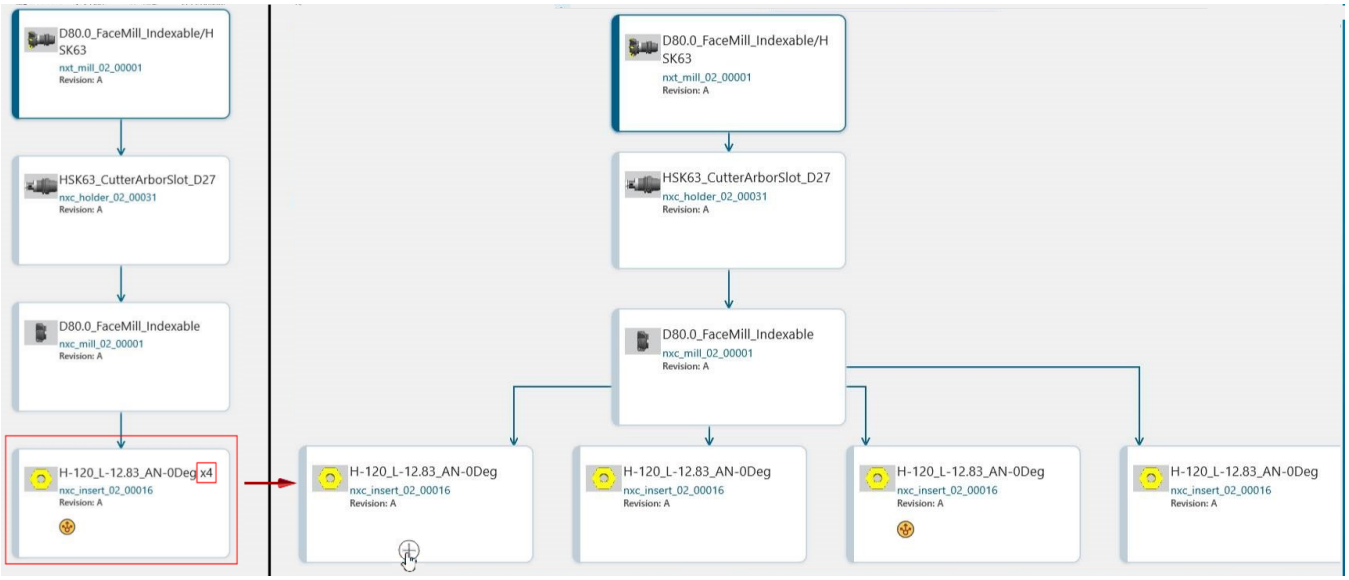
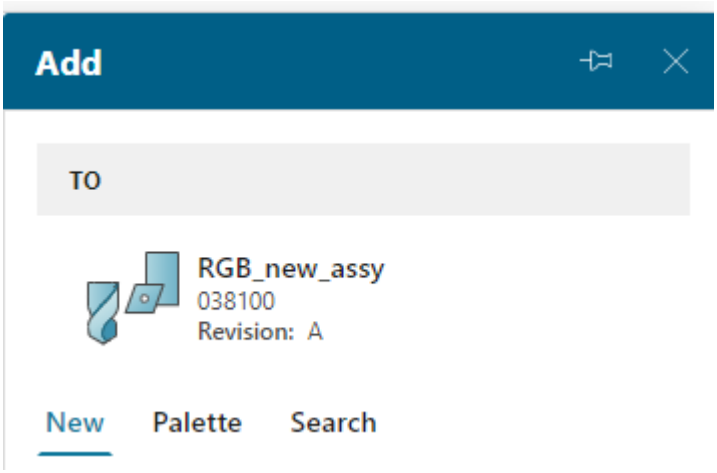


Add new components to a tool resource assembly

Start with an existing resource or create a new one.

1. Select **Add** ⊕ →**Add Resource Component** ⊕ on the work area toolbar.

2. Select the **New** tab.



3. Search for **NC Tool** in the **Filter** field or select it in the **Recent** list if available.

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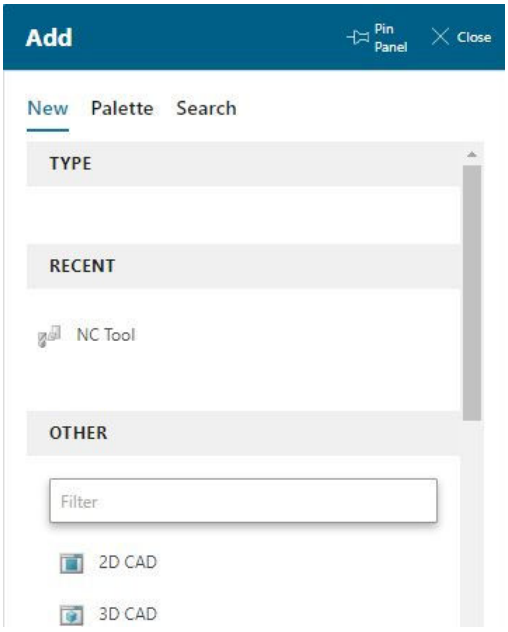

4. Enter a name for the new component in the **Name:** field.

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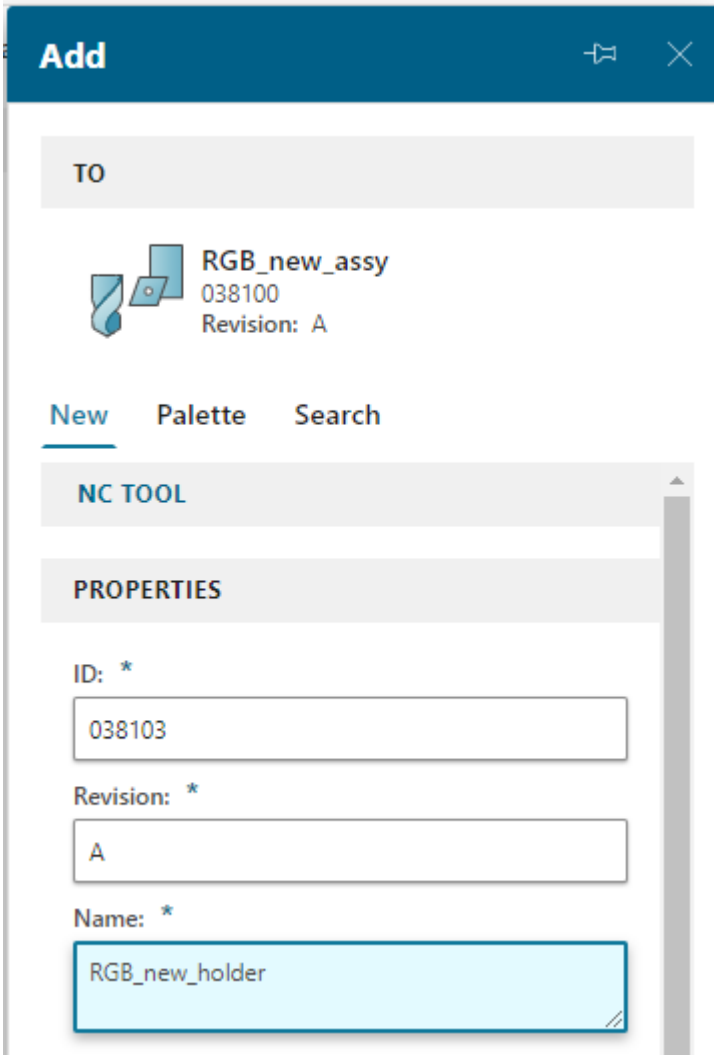

5. Select **Add**.

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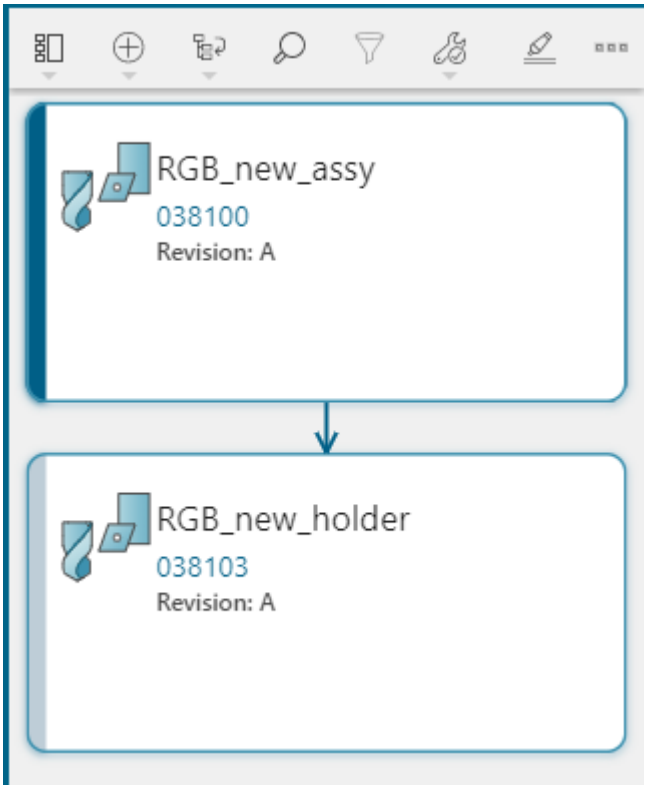

Note:

After you add a new component, you can classify it by selecting the **Classification** tile from the Active Workspace home page.

For more information on classifying resources, see Active Workspace, Advanced Classification — Deployment and Administration.

Add existing NC Tool resource component

1. Open or create an **NC Tool** resource that becomes the root node.
2. Select the **Add Resource Component** ⊕ icon from the menu bar above the summary view or from one of the nodes in the assembly.
3. Click the **Search** tab in the **Add** dialog box.
4. Enter information into the **Search** input field to begin your search.
5. Find the tool you need in the **Results** list or use **Filters** to narrow the search results.

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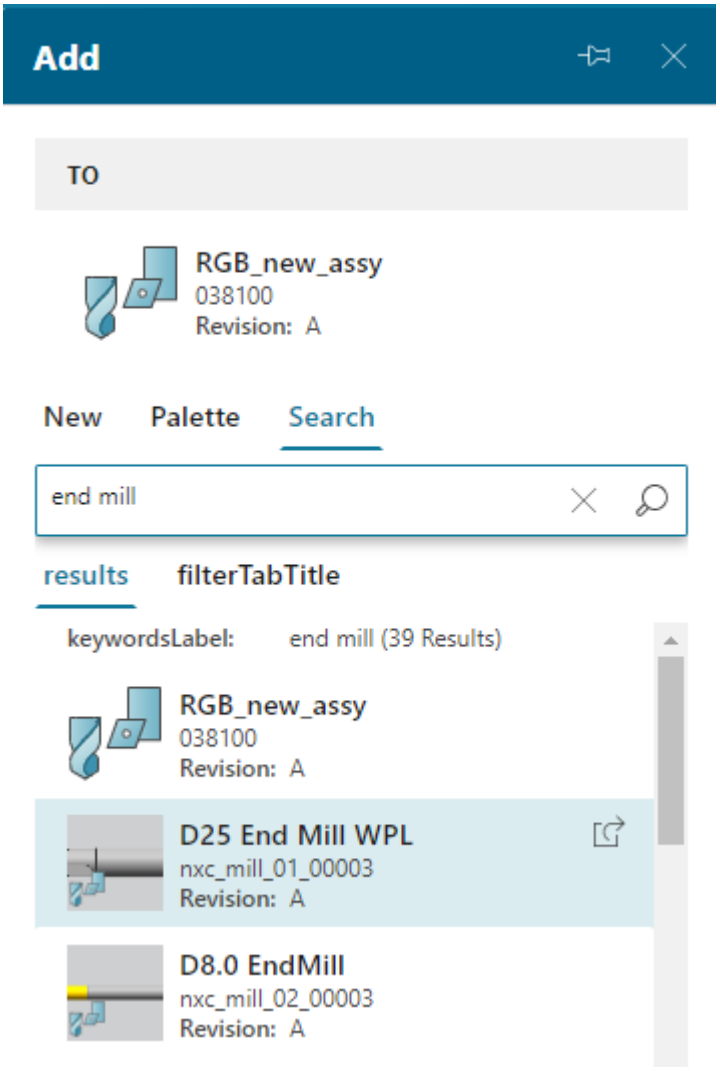

6. Once you locate the existing tool you want to add, select **Add**.

Add existing component by copy and paste

1. Use **Search** to locate an existing component to add to a tool assembly.

2. Select the component and select **Copy** from the **One Click Command** area on the right side of the page.

3. Navigate to the tooling assembly to which you want to add the component.

4. Select the node at the appropriate level in the structure.

5. Select **Paste** from the **One Click Command** area on the right side of the page.

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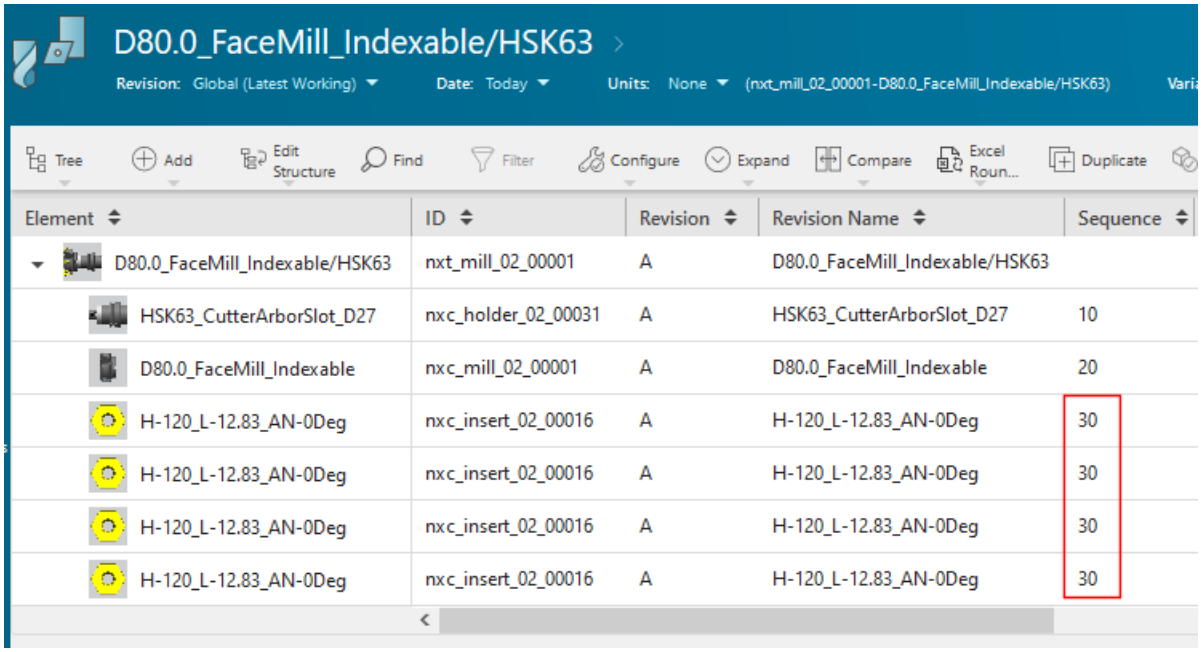
Assigning components the same sequence number

You can navigate a tooling assembly easily, because Resource Manager assigns the same sequence number to each component when you add two or more identical components to the assembly; for example, components such as cutting inserts. Having the same sequence number means that the components are packed into a single node in the resource diagram, which makes the assembly easier to navigate.

> Note:
>
> The sequence number is also assigned automatically when you add a single component.

- When you add two or more components with the same ID to a tooling assembly, they have the same sequence number.



- When you add additional component to an assembly that already contains a component with the same ID, the new components have the same sequence number.

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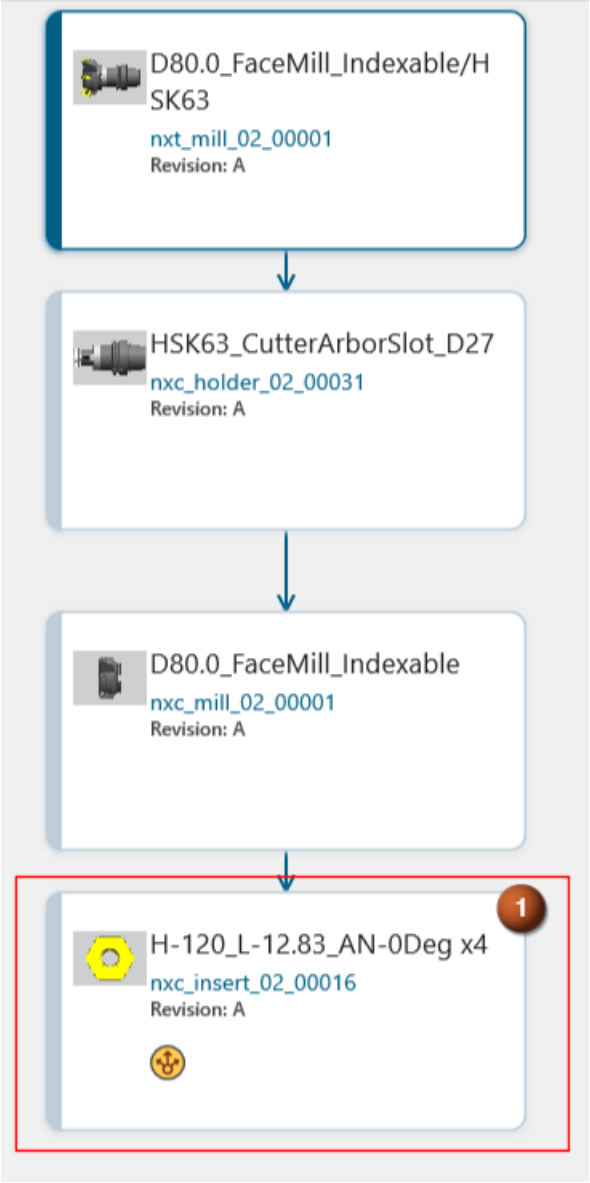

- When you add two or more components with the same ID, the component nodes in the **Resource with Summary** and **Resource** diagrams are automatically packed and the quantity is noted. In the example (1), the quantity is noted in the packed node by **x4** after the name.

- If you already have multiple components with the same ID in the assembly, and the nodes are expanded when you add the additional components, the new nodes are not packed but have the same sequence number.

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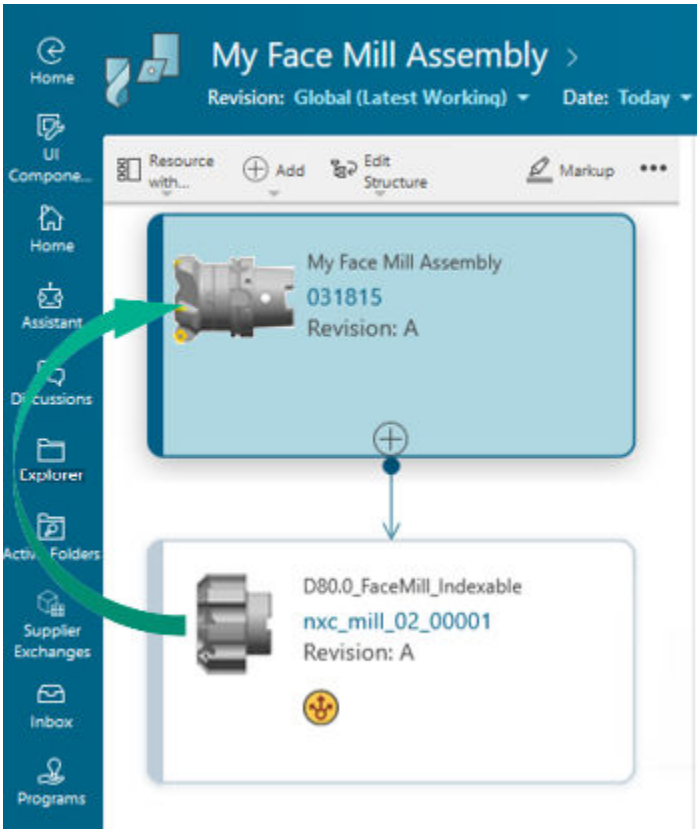
Creating resource assemblies leveraging resource information

Using existing information for new resource assembly

Use **Create Parent Assembly** to quickly build a new tool assembly that is based on an existing cutting tool component.



When you use **Create Parent Assembly** to create a new tool assembly:

- The tool assembly is automatically classified in the correct tool assembly class.
- The selected tool component is added as a child to the tool assembly structure.
- The propagation start point is assigned to the cutting component.
- The tool assembly is saved to the database. The cutter attribute values, such as the cutting diameter, corner radius, or tool material, are visible at the assembly level.

Create a parent assembly from a component

Use **Create Parent Assembly** to create a tool assembly that is automatically classified, based on the selected component.

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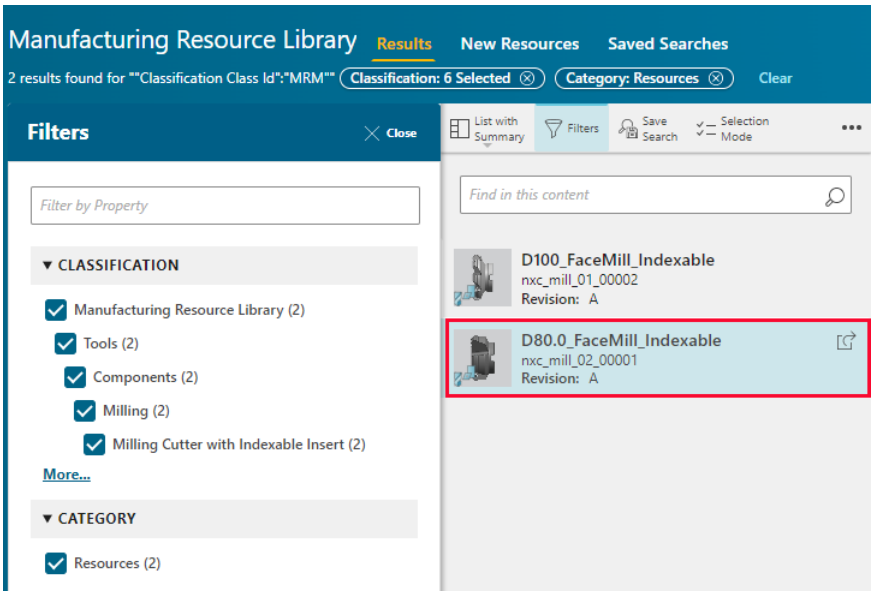
1. On Active Workspace, click the Manufacturing Resource Library tile.

> Note:
>
> This feature is only available for drilling, milling, and turning tool components.

The Manufacturing Resource Library (MRL) dashboard opens.



2. In the MRL dashboard, search for a tool component and select it from the results panel.

3. Select **More commands** •••→**New** ✧→**Create Parent Assembly** on the primary tool bar.

The **Create Parent Assembly** dialog box opens.

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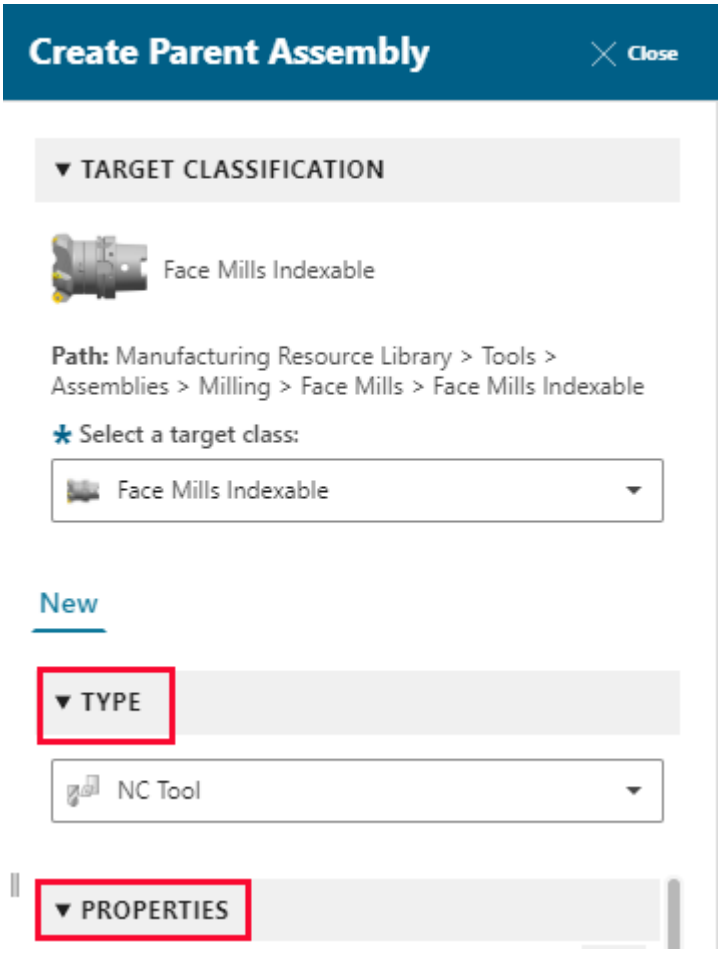

4. In the **Create Parent Assembly** dialog box, select the **TYPE** and enter the **PROPERTIES**.

ID	MRL automatically assigns a unique ID that you can accept or change. If you change it, the new ID must be unique.
Revision	If this is a revision of an existing resource, type the next consecutive revision letter.
Name	Type the name of the new resource. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the new resource.

5. Click **Create**.

   A message box appears to confirm the tool assembly is created.

Revise resources and parent assemblies

Use **Revise Including Parent Assemblies** to revise a resource and all parent assemblies.

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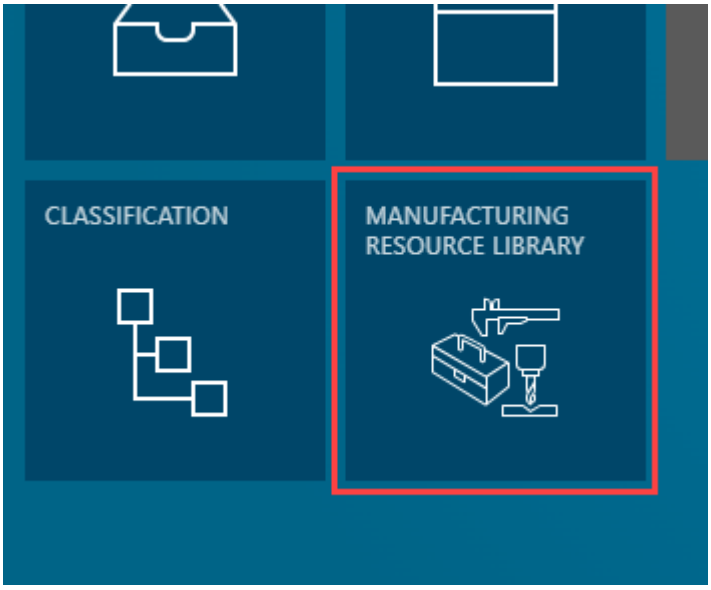
Procedure

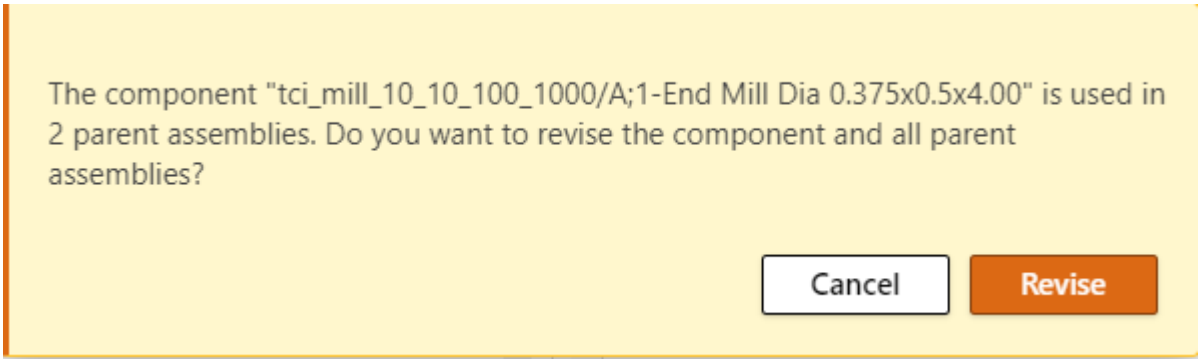
1. On Active Workspace, click the Manufacturing Resource Library (MRL) tile.



2. On the Manufacturing Resource Library (MRL) workspace select the resource to revise.

3. Select **More commands** •••→**New** ✳→**Revise Including Parent Assemblies** on the primary toolbar.

   A message shows the number of parent assemblies associated with the selected component.



4. Click **Revise**.

Searching for matching components

You can build a tooling assembly easier and faster in Active Workspace by using the information on the **Matching Components** tab. This information allows you to choose from a list of classified objects that fit with the selected component.

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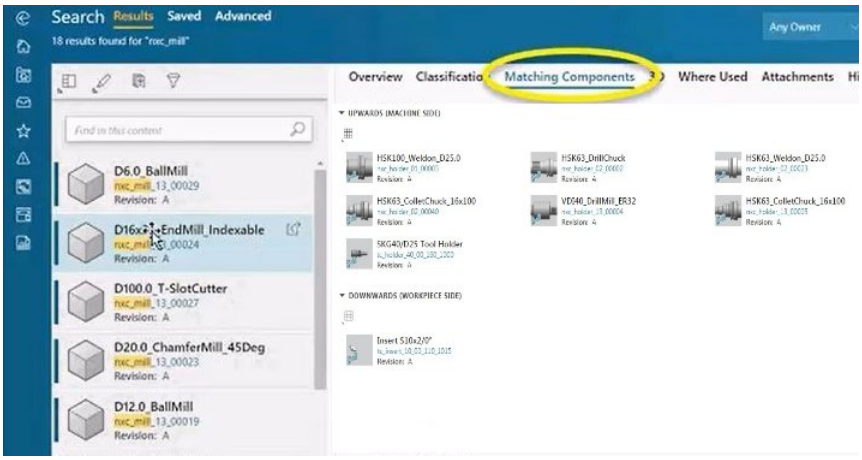
Note:

The available tabs and their order can differ depending on what templates were imported and where you are in Active Workspace.

Example:

In the **Search** results, select **D16x35 EndMill_Indexable** and select the **Matching Components** tab. The components that can be used with the end mill are grouped by components that attach **Upwards**, or closer to the machine, and components that attach **Downwards**, or closer to the workpiece.



The **Matching Components** tab is available for item type **Resource** and for **Resource** subtypes.

Note:

The template **Manufacturing Resource Manager (MRL)** must be installed in order for the **Matching Components** tab become available. It is installed using the Teamcenter Environment Manager and is found under **Server extensions**.

Connection types (CT) and **connection point definitions (CPD)** must be imported in order to take advantage of **Matching Components**. These rules are delivered with MRL and can be imported using one of two options:

- **Import rules and preferences**
- **Guided Component Search rules for tool components**

You access the **Matching Components** tab from the **Home** folder, from the **Search** results page, from the **Item Revision** page, and from the assembly content page.

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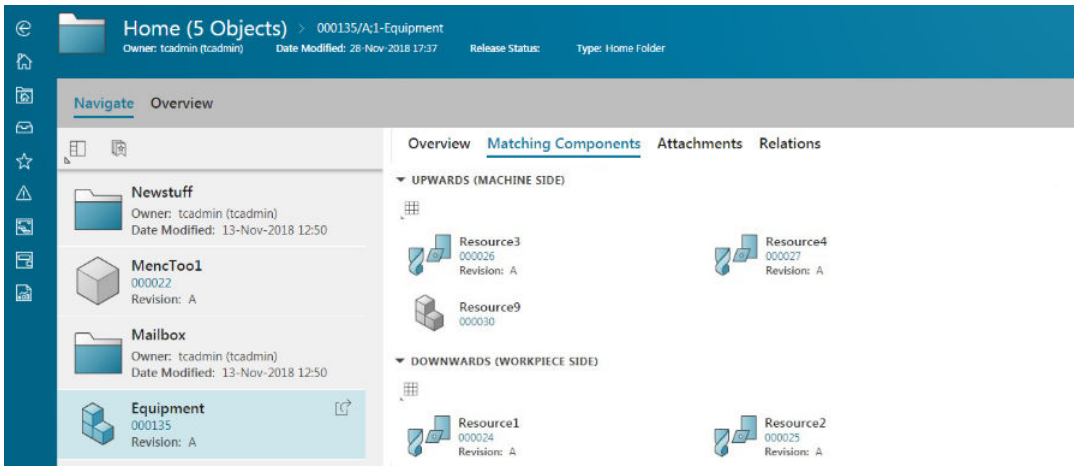
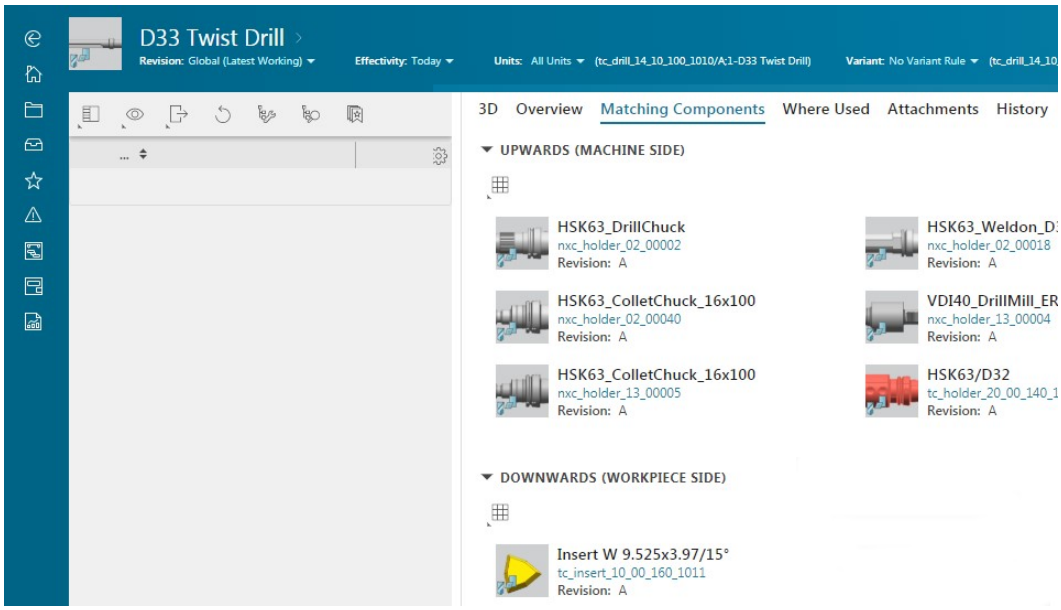
Matching Components from the Home page



Matching Components from the Item Revision page



Matching Components from the assembly content page

You can select the assembly from the assembly content page.

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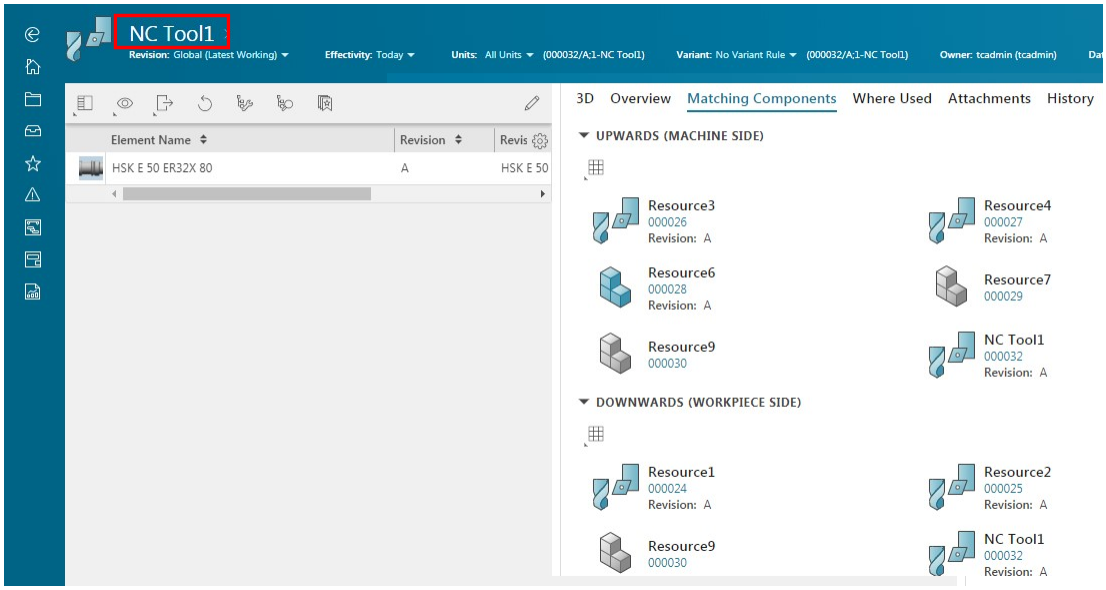
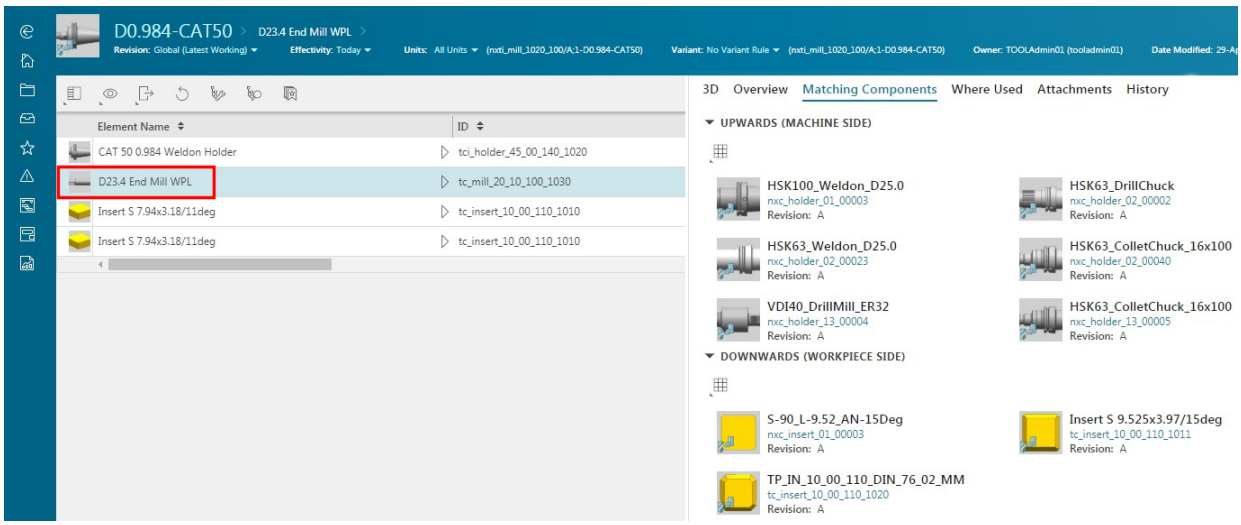


You can select a component from the assembly content page.



Add matching component to a tool assembly

1. Start with an existing resource assembly or create a new one.

   > Note:
   >
   > The assembly must have at least one component.

2. Select a component of the assembly and click the **Matching Components** tab.

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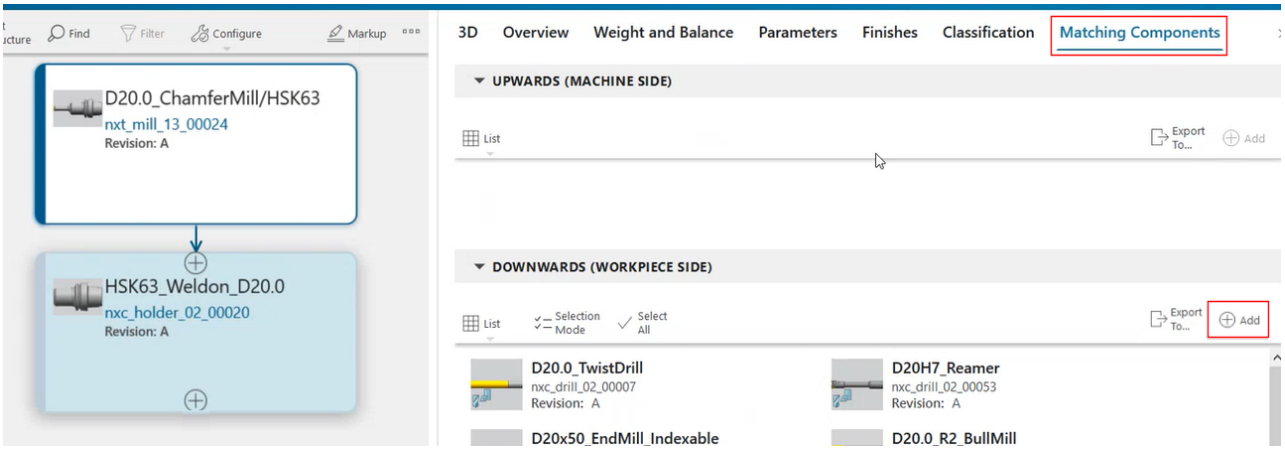
Note:

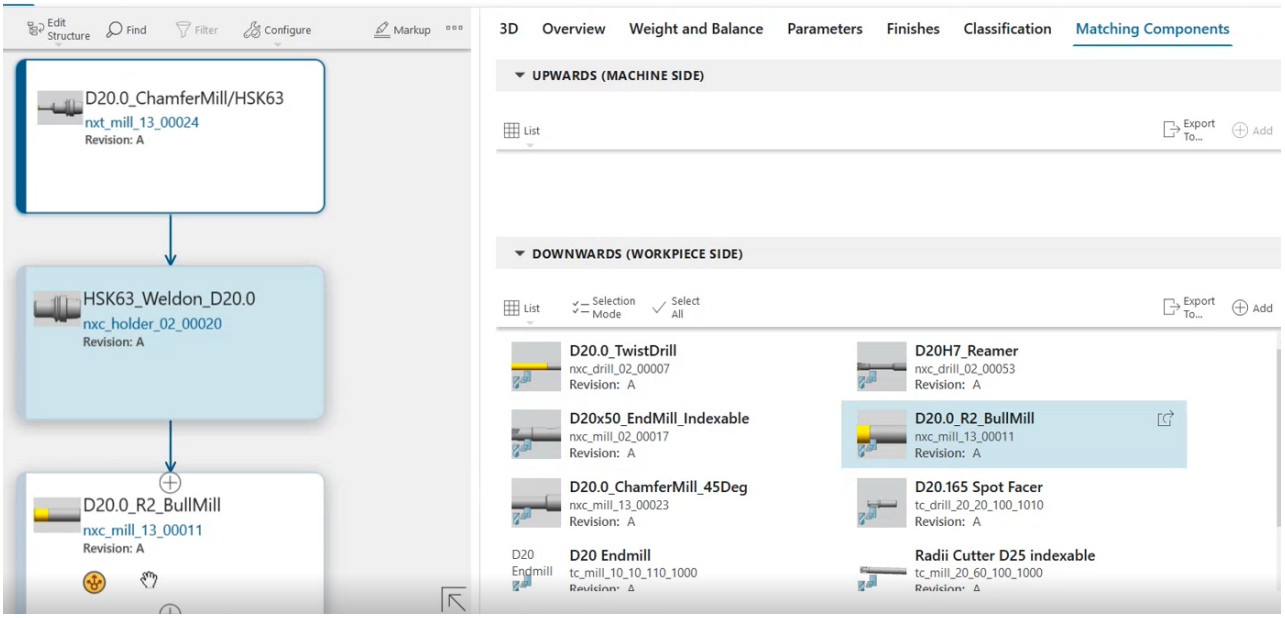
The available tabs and their order can differ depending on what templates were imported and where you are in Active Workspace.



In this example, there are no matching components in the upwards direction, or the machine side of the selected component. There are several components that do match the selected component in the downwards direction, or the workpiece side.

3. Select a matching component from the **UPWARDS (MACHINE SIDE)** section or the **DOWNWARDS (WORKPIECE SIDE)** section and click ⊕ **Add**.

In the example, the bull mill is selected and added to the assembly below the selected component, on the workpiece side.



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Add multiple matching components to a tool assembly

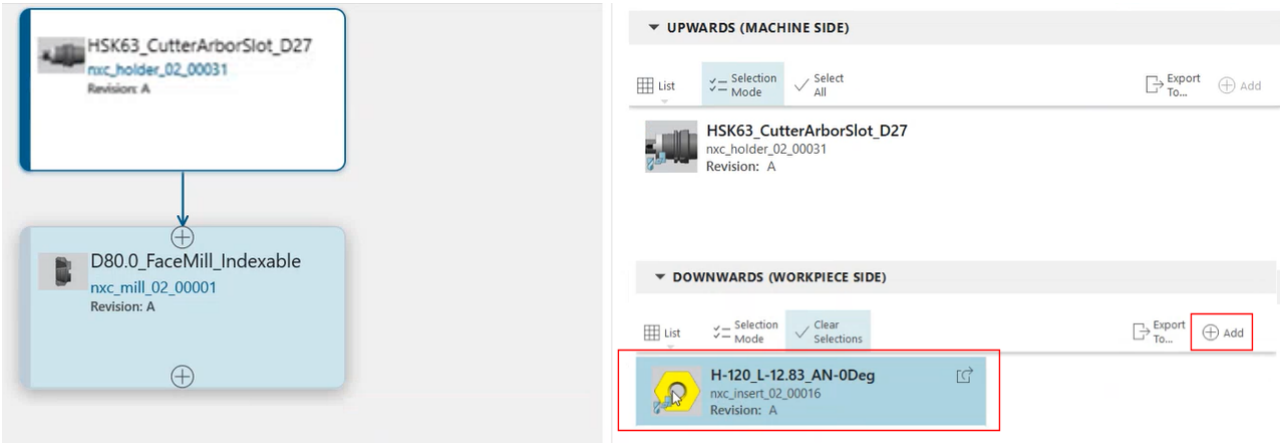
1. Open an existing resource assembly or create a new one.

> Note:
>
> The assembly must have at least one component.

2. Select a component of the assembly and click the **Matching Components** tab.

> Note:
>
> If the selected component has multiple matching components, such as a milling cutter with multiple cutters, you are prompted to define or confirm the number of matching components you are adding.

3. Select the matching component, in this example the **H-120_L-12.83_AN-0Deg** cutter.



4. Click ⊕ **Add**.

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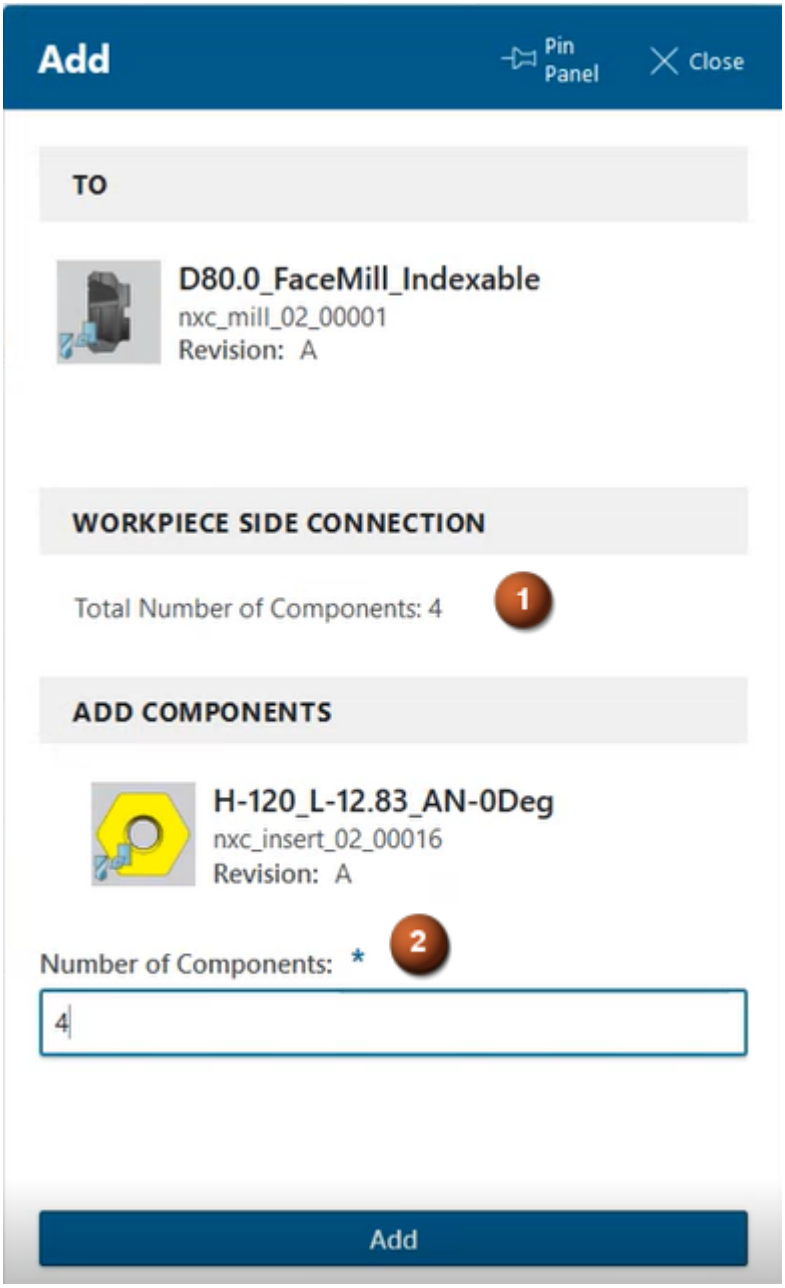

Because there is the possibility of adding multiple cutters, the **Add** dialog box displays additional information regarding the number of cutters.

- The maximum number of cutters for this indexable face mill (1).
- The number of cutters you specify. The default value is the number of components for which there are available locations (2).

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> Note:
>
> If you have already added two cutters and are adding more, and the maximum number of cutters is four, the default is two, based on the remaining available locations.

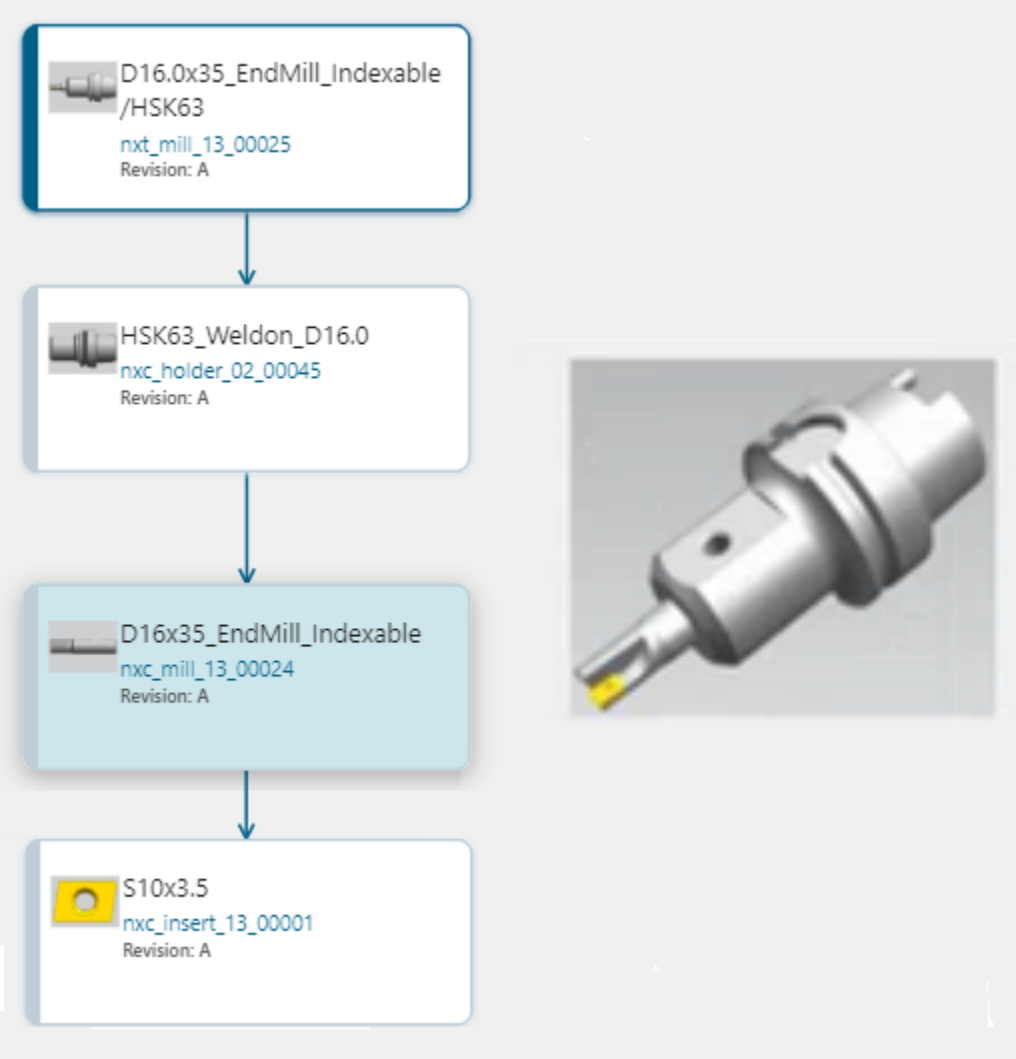
5. Accept the default number of components or enter a smaller number.

6. Click **Add**.

Positioning components automatically

Auto-positioning when you add a single component

When you add a component to the resource structure, the system automatically positions the 3D geometry according to the **Coordinate System Workpiece Side (CSW)** and **Machine Side Coordinate System (MCS)** of the component selected in the structure.



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Note:

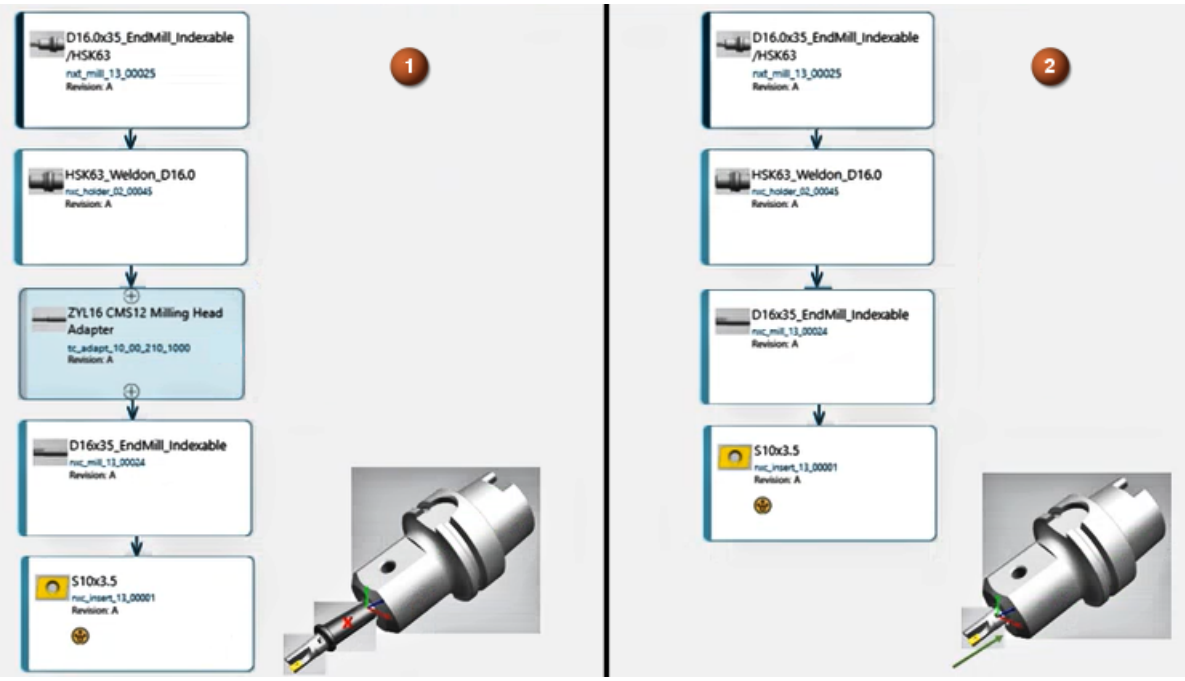
If there is only one **CSW** present, the component is automatically placed relative to that coordinate system. If two or more **CSWs** are present, the component is added to the first open position.

Auto-positioning when you add multiple components

When you add two or more components to the resource structure and the mating component has two or more **CSWs**, the system also automatically positions the 3D geometry based on related coordinate systems, beginning with the first open position.

Auto-positioning components when you add, remove, or replace a component

When you add, remove, or replace components that have other components attached to them, the system automatically positions the other components.



In the example above, the component **ZYL16CMS12 Milling Head Adapter** is removed (1). The component **D16x35_EndMill_Indexable** and the cutters attached to it are properly repositioned (2).

The system also automatically handles the positioning if you insert a component that requires a mating component to move to a different location. For example, if you insert the adapter between the chuck and the end mill, the system automatically handles the positioning of the components.

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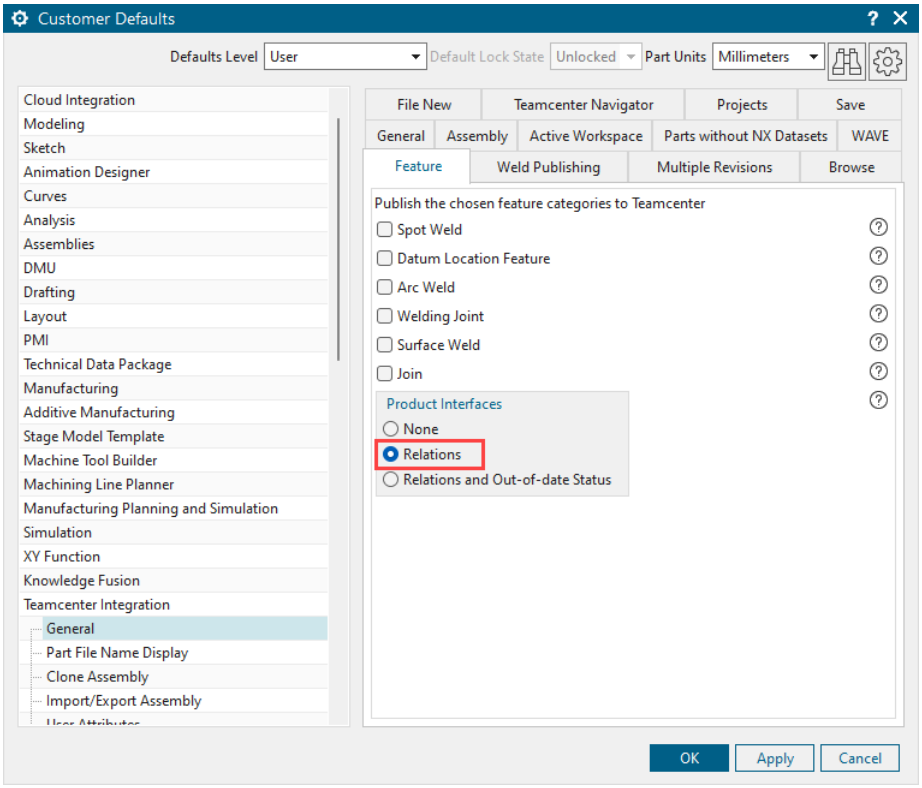
Tip:

Confirm the auto-position setting in NX: **File→Utilities→Customer Defaults→Teamcenter Integration→General→Feature→Product Interfaces** is set to **Relations**.



Create and publish a coordinate system in NX

If you use the Manufacturing Resource Library, the coordinate systems for each component are defined in the part family templates. In a manual configuration, you must perform the following steps to publish the coordinate systems from NX to Teamcenter:

1. In NX, create a coordinate system of the **Datum CSYS** type and give it a name that conforms to the DIN4003 naming standard.

2. Add the coordinate system to the product interfaces by choosing **Tools→Product Interface**.

3. Choose **File→Options→Save Options** and select **Save JT Data**.

Tip:

Confirm the auto-position setting in NX: **File→Utilities→Customer Defaults→Teamcenter Integration→General→Feature→Product Interfaces** is set to **Relations**.

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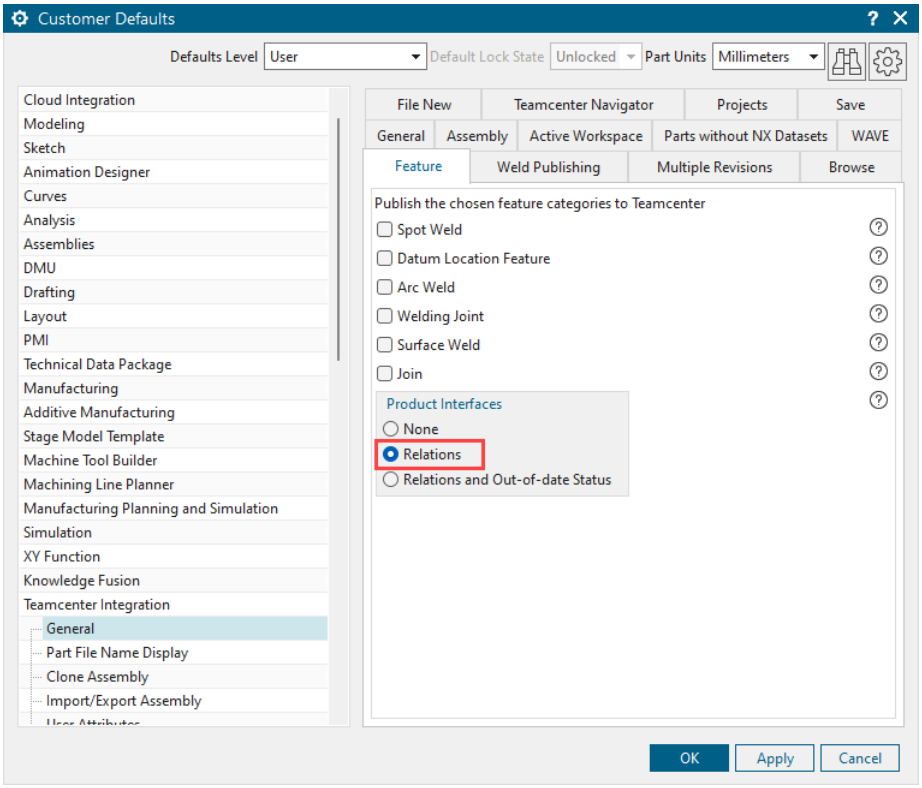

DIN4003 coordinate system naming scheme

The DIN4003 standard defines how 3D graphic models for different tool component classes must be designed. This standard also defines the coordinate systems in the tool components using the following naming scheme:

MCS

Machine-side coordinate system

CSW

Coordinate system workpiece-side

A tool component always has one MCS and any number of CSWs, depending on its design. If it has a single CSW, the coordinate system is called **CSW**. If it has multiple CSWs and dedicated steps (see example), the coordinate systems are additionally numbered in ascending order, using a system that is dependent on the shape of the component. The examples provide an introduction to the naming convention.

Tool component	MCS	CSW
Insert: **DIN4003_76_01 [TC_INSERT_10_00_100]**	Pointing toward the holder	None

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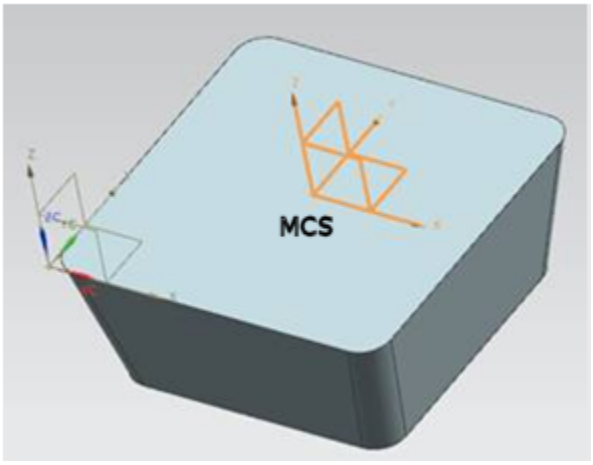
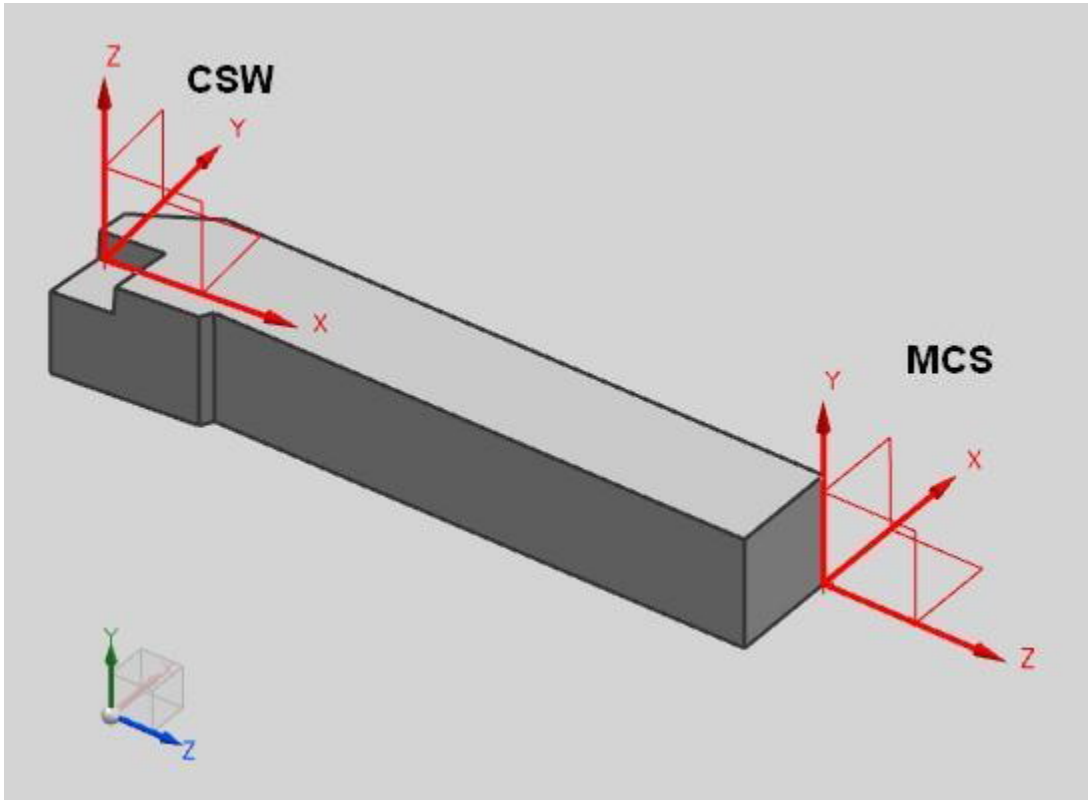


Tool component	MCS	CSW
Holder: **DIN4003_90_02 [TC_TURN_40_00_100]**	Pointing toward the machine adapter	**CSW** pointing toward the insert



Tool component	MCS	CSW
Drill: **DIN4003_86_04 [TC_DRILL_20_20_100]**	Pointing toward the machine adapter	**CSW_1** pointing toward the insert **CSW_2** pointing toward the insert

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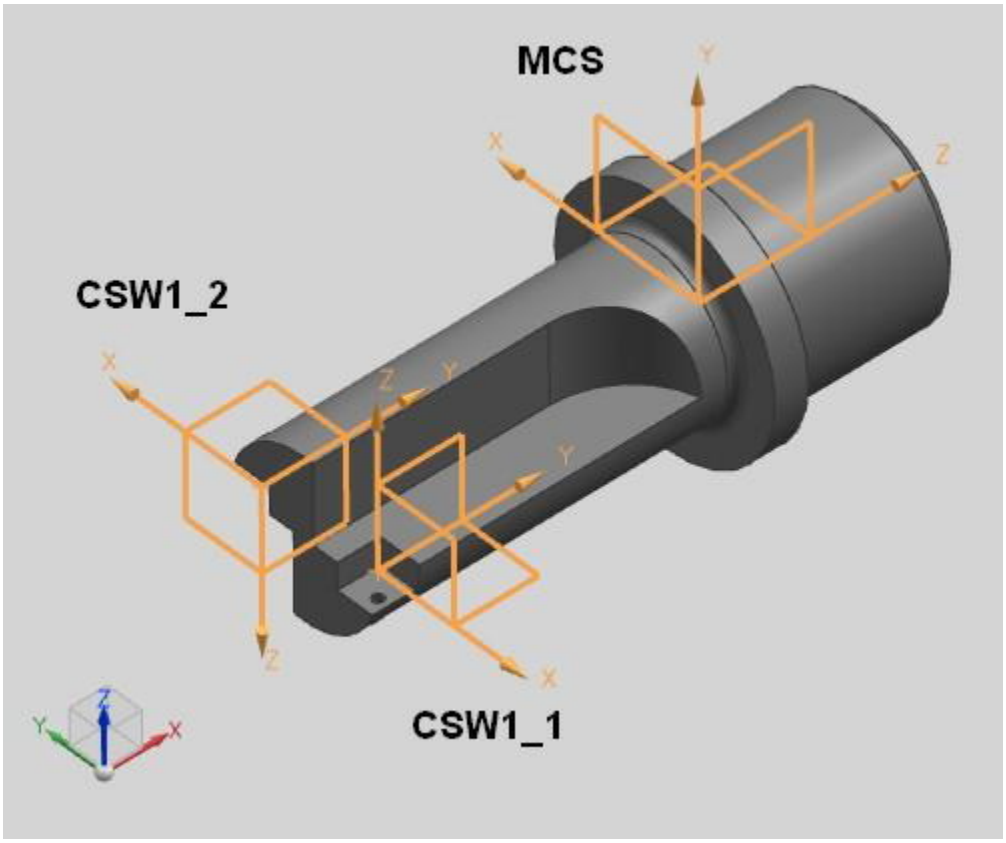

Tool component	MCS	CSW
Step drill: **DIN4003_86_05 [TC_DRILL_20_20_110]**	Pointing toward the machine adapter	First step: **CSW1_1, CSW1_2** (pointing toward the inserts) Second step: **CSW2_1**, **CSW2_2** (pointing toward the inserts) Third step: **CSW3_1**, **CSW3_2** (pointing toward the inserts) Fourth step: **CSW4_1**, **CSW4_2** (pointing toward the inserts)

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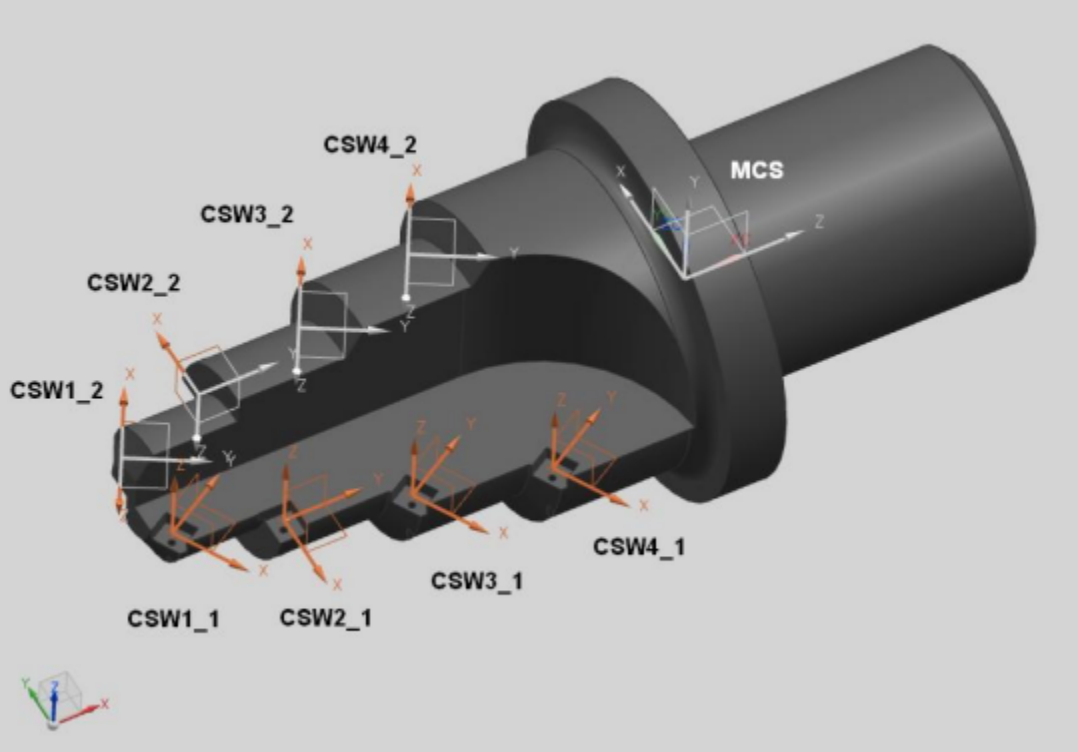

Tool component	MCS	CSW
Shell end mill: **[TC_MILL_20_20_120]**	Pointing toward the machine adapter	**CSW_1**, **CSW_2**, **CSW_3**, **CSW_4**, **CSW_5**, **CSW_6**, **CSW_7**, **CSW_8** (all pointing toward the inserts)

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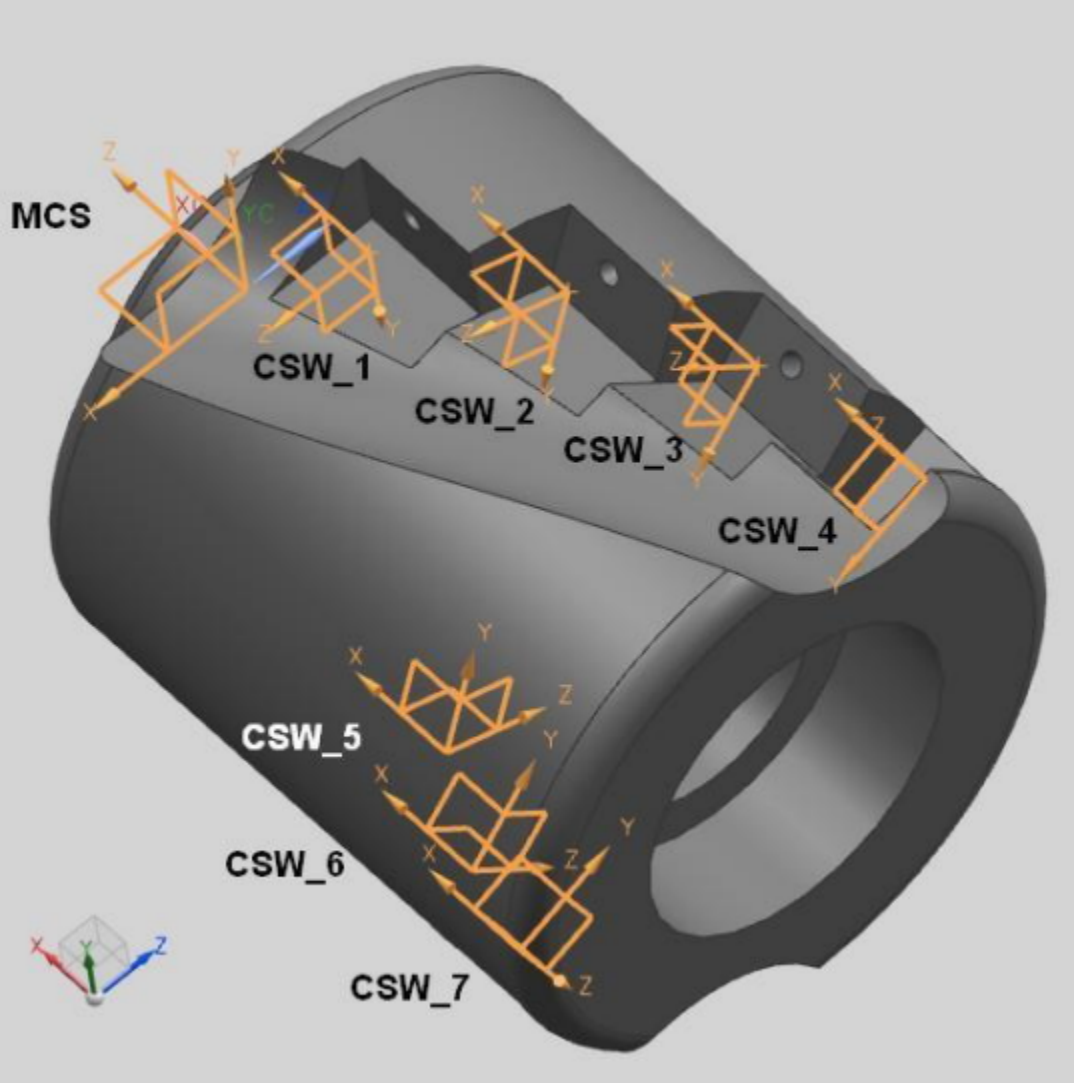

Tip:

Confirm the auto-position setting in NX: **File→Utilities→Customer Defaults→Teamcenter Integration→General→Feature→Product Interfaces** is set to **Relations**.

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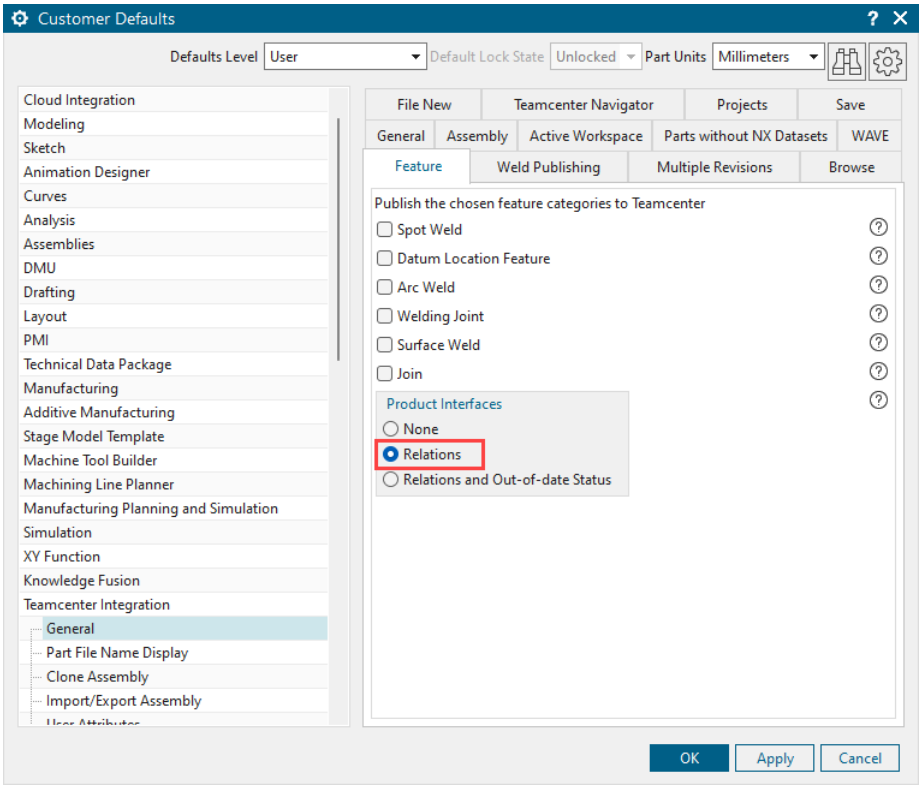
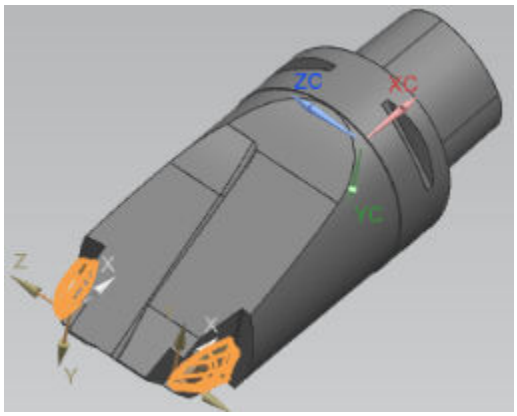
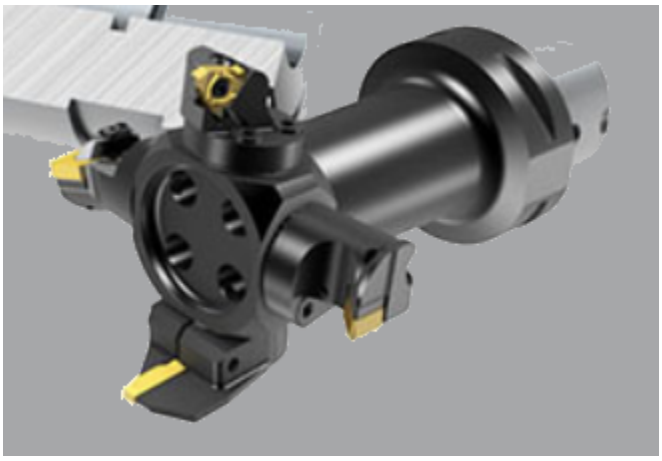


Working with multitool assemblies

Overview of multitool assemblies

Multitools are tool assemblies that have more than one cutter.





Multitools save costs by reducing the need for extra tool components, and save setup time between operations that use different cutters. Using Resource Manager, you can create multitools in a way similar to the methods for creating other tools. Resource Manager displays the multiple cutters in one assembly.

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Each cutter is assigned its own tool type and is able to have its own attribute propagation start point (PSP) in the resource bill of materials (BOM) structure.

You can also search specifically for multitools in NX CAM and retrieve them from the Manufacturing Resource Library (MRL). The required tool parameters are passed from Resource Browser to each cutter in the multitool assembly. The multitool is then used to create and simulate the tool path.

Multitool classification

Multitools are stored in the existing tool assembly classes. In support of multitools, the following multitool attributes are available in the assembly classes:

- **Multitool [MT]**—Identifies a multitool
- **Cutter ID [CID]**—Unique cutter identifier used for multitools
- **Cutter Name [CNA]**—Description for a cutter typically used in a multitool
- **Index Notch [INN]**—Rotation angle around tool axis to position tool cutter
- **Turret Rotation [TUR]**—Incremental turret rotation angle to position tool cutter

> Note:
>
> The attributes **Multitool [MT]** and **Cutter ID [CID]** are set to read-only. Resource Manager automatically updates these attributes.

How Resource Manager handles cutter IDs for multitools

Resource Manager manages multiple cutters by assigning unique cutter IDs for each cutter in the multitool assembly.

> Example:
>
> The multitool with the item revision ID **nxt_multi_04_00815/B** has three cutters assigned, with these ID classification objects (ICOs):
>
> - **nxt_multi_04_00815/B#1**
> - **nxt_multi_04_00815/B#2**
> - **nxt_multi_04_00815/B#3**

When you create multitools, the system first checks if the existing tool has multitool classification objects attached, and determines the appropriate cutter ID. The default cutter ID is **2**, unless more than one

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cutter is already in the assembly. If more than one cutter is present, the next cutter added is assigned the next available cutter ID number. The assigned cutter ID is stored in the attribute **Cutter ID [CID]**.

ICO assignment for multitools

When you create a multitool assembly, a new ICO is created and attached to the same classified tool item or revision as the original tool assembly.

By default, the new ICO resides in the same class as the original tool assembly. All shared values are copied into the new cutter ICO, while all other fields remain initially empty. One exception is the **NX-Tool-Type**, which is initialized according to the default value for this class.

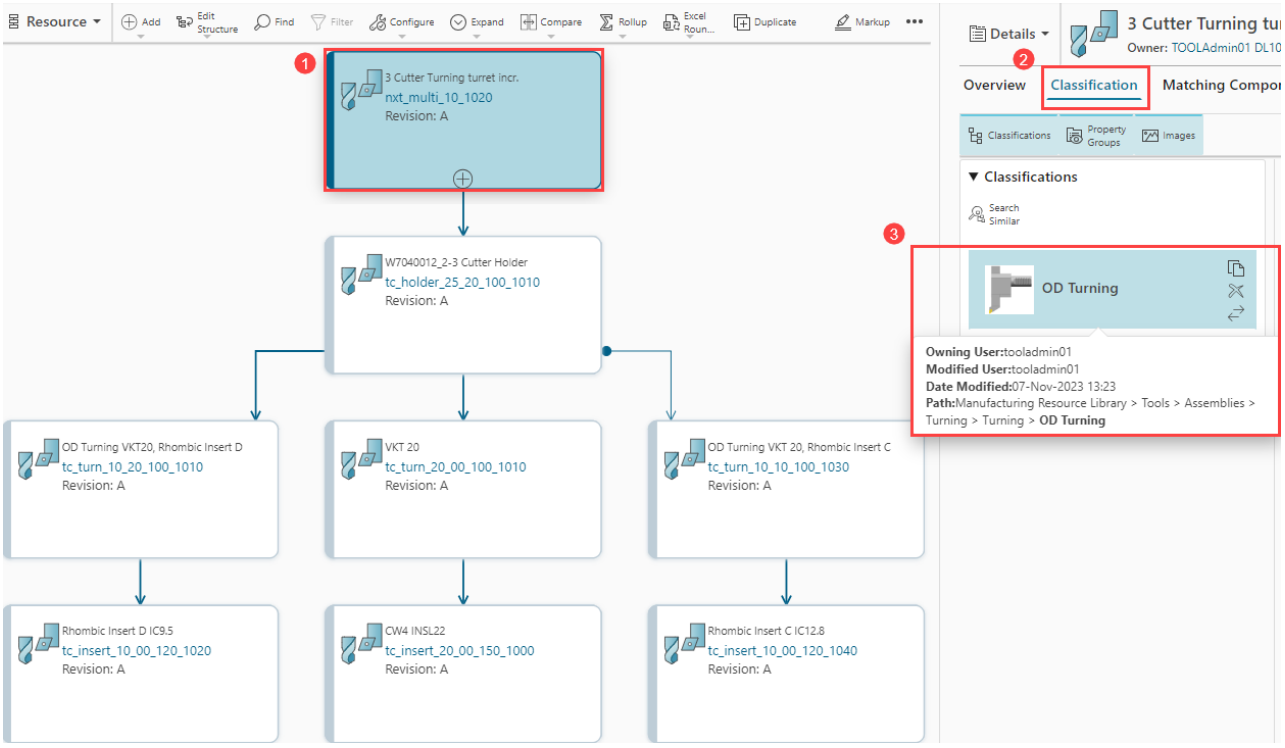
> Note:
>
> The above condition results in tools having multiple classifications defined to attach cutter ICOs. We do not recommend that you use *multiple classification* with tool assemblies in other scenarios.

Create a multitool assembly

Create a multitool assembly from a tool assembly that includes two or more cutters.

1. Open an existing tool assembly, or if needed, assemble a new tool assembly that contains multiple cutters.

2. (Optional) Add additional cutters to create the multitool.



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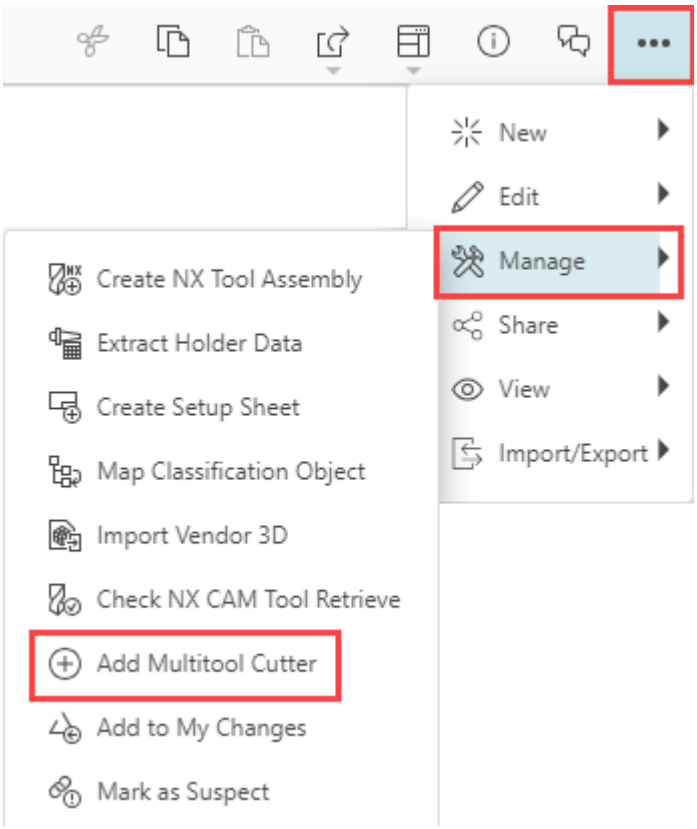
(1)—Root node

(2)—**Classification** tab

(3)—Hover text that shows additional cutter attributes

> Tip:
>
> Hover over a cutter classification to display additional cutter attributes. This can help you identify the cutter you want to select.

a. Select the root node and select **More commands** •••→**Manage** 🛠→**Add Multitool Cutter** ⊕.



A message confirms that a cutter is added.

b. Select the **Classification** tab and verify the classification attributes for the cutters.

- You can confirm the multitool has been created when two classifications are present.

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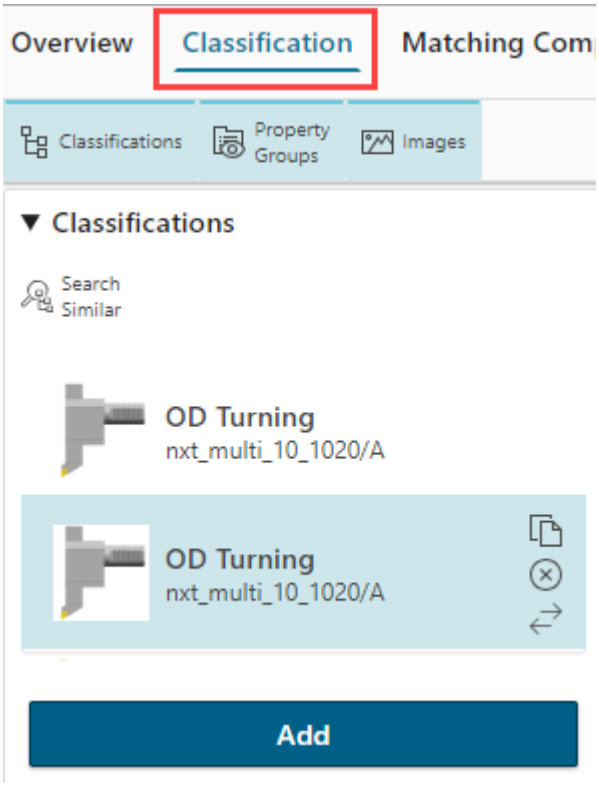

- When a cutter is added, Resource Manager automatically updates the status and cutter ID attributes in the **Multitool Data** group, such as **Multitool [MT]: 1 Yes** and **Cutter ID [CID]: 1**.

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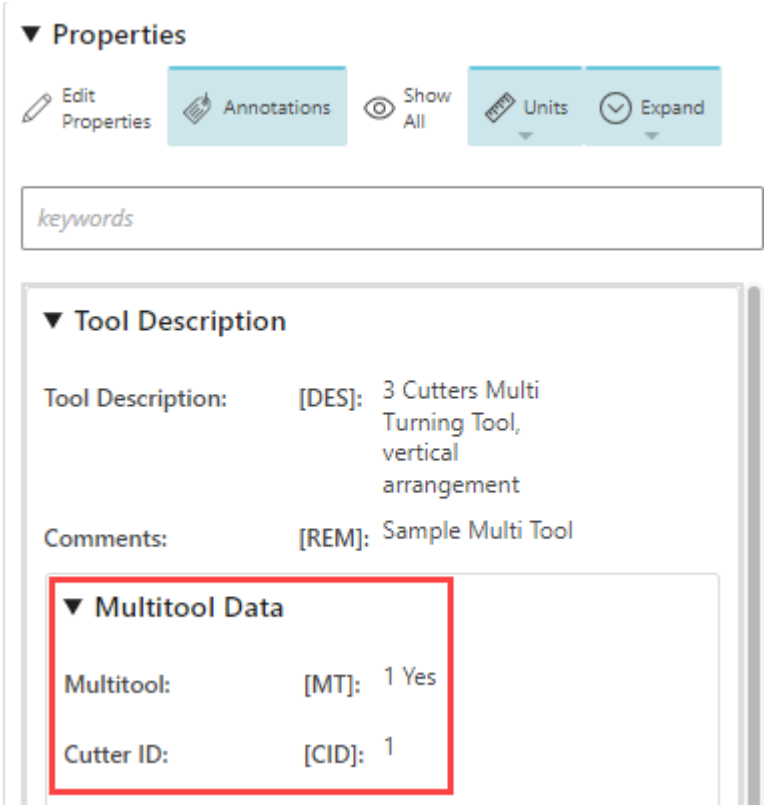
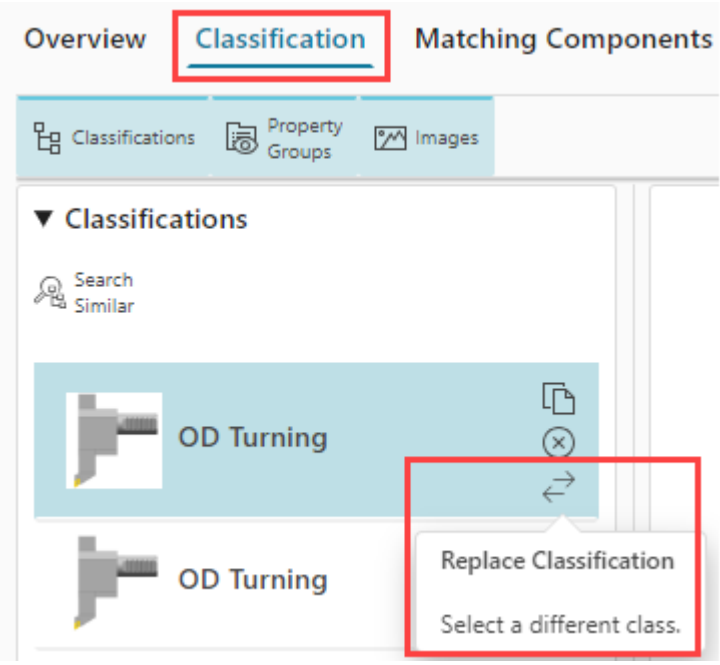


3. (Optional) Adjust the classification if needed, or if the cutter uses a different tool type.

   a. On the **Classification** tab, click **Replace Classification** ⇄ next to the specific cutter.



   b. Select the needed tool assembly classification in the **Classifications** group and click **Classify**.

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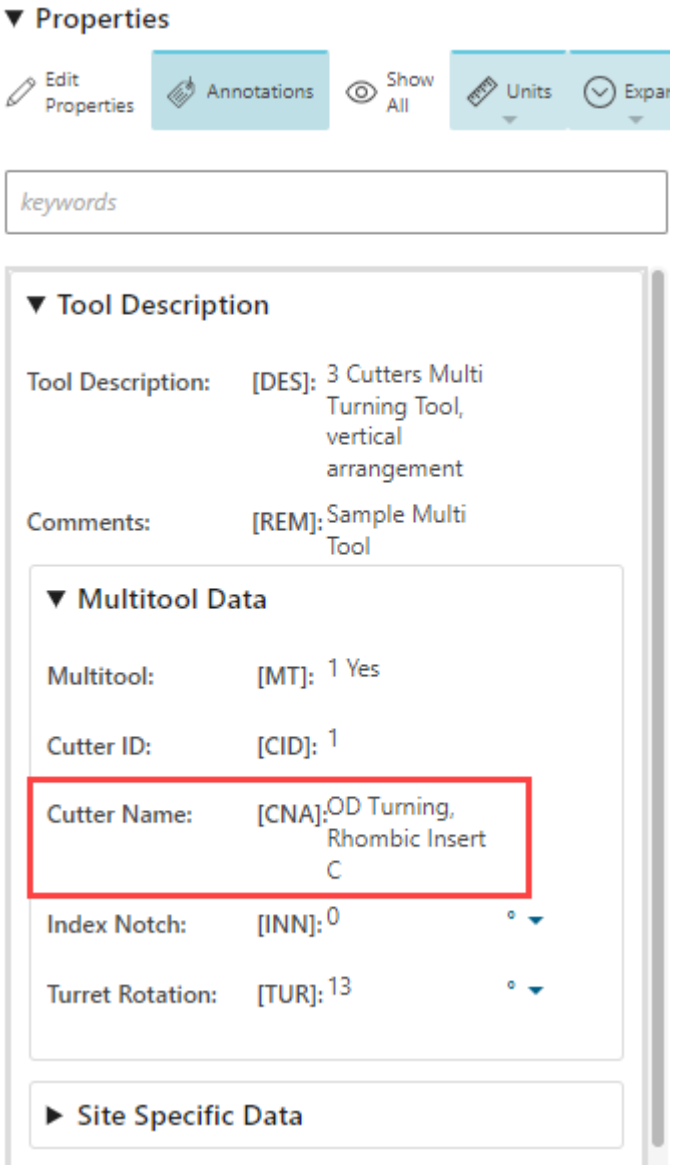
4. Enter or adjust the cutter name for each cutter to define tool-specific attribute values.

   Select **Properties→Tool Description→Multitool Data→Cutter Name [CNA]**, make changes, and save.



5. (Optional) Adjust the tool angles for each cutter.

   Select **Edit Properties** to change or assign other cutter attributes, such as **Index Notch [INN]** or **Turret Rotation [TUR]**, make changes, and save.

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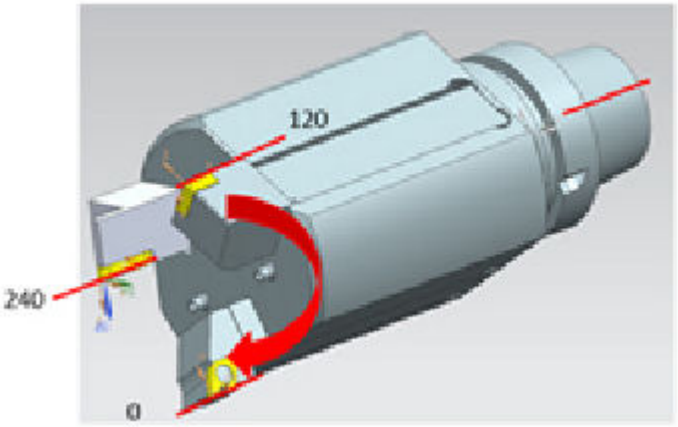
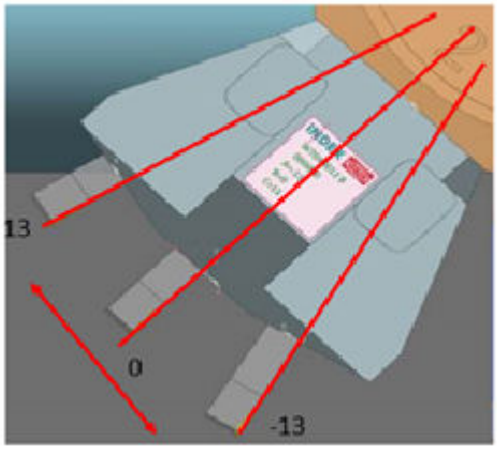


Index Notch [INN]



Turret Rotation [TUR]

6. Set the propagation start point (PSP) for each cutter.

> Note:
>
> To propagate the correct classification attributes from the insert components to each cutter at the tool assembly, a propagation start point must be assigned.
>
> For each cutter, the attributes are propagated from the components to the tool assembly.

a. Select the cutter in the resource structure and select **More commands •••→Manage →Propagation Start Point (PSP)** .

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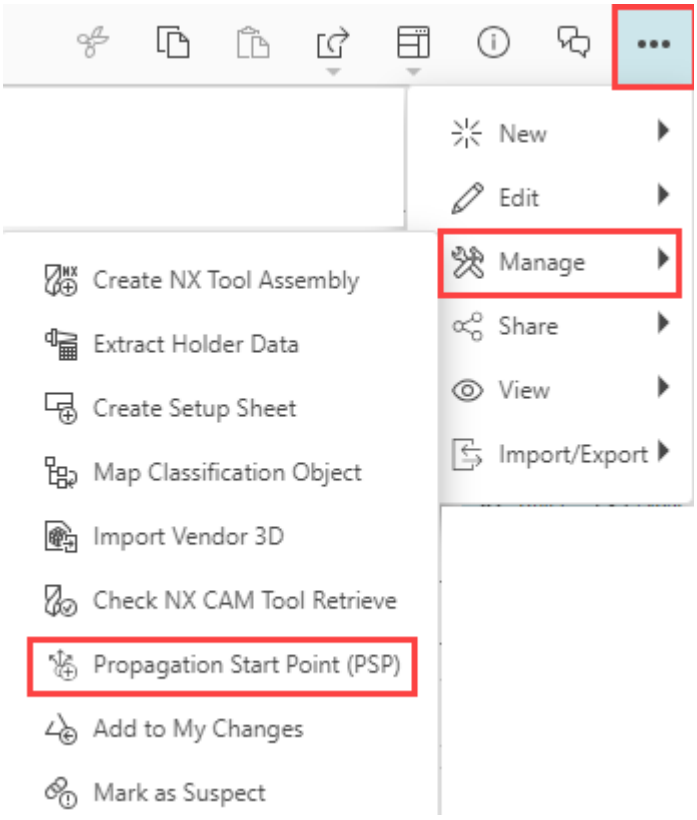

b. In the **Propagation Start Point** dialog box, select a cutter, and click **Add**.

> Note:
>
> For the cutter that is selected, the color of the propagation start point icon changes from gray to yellow.

> Tip:
>
> Hover over a cutter in the **Propagation Start Point** dialog box to display additional cutter attributes.

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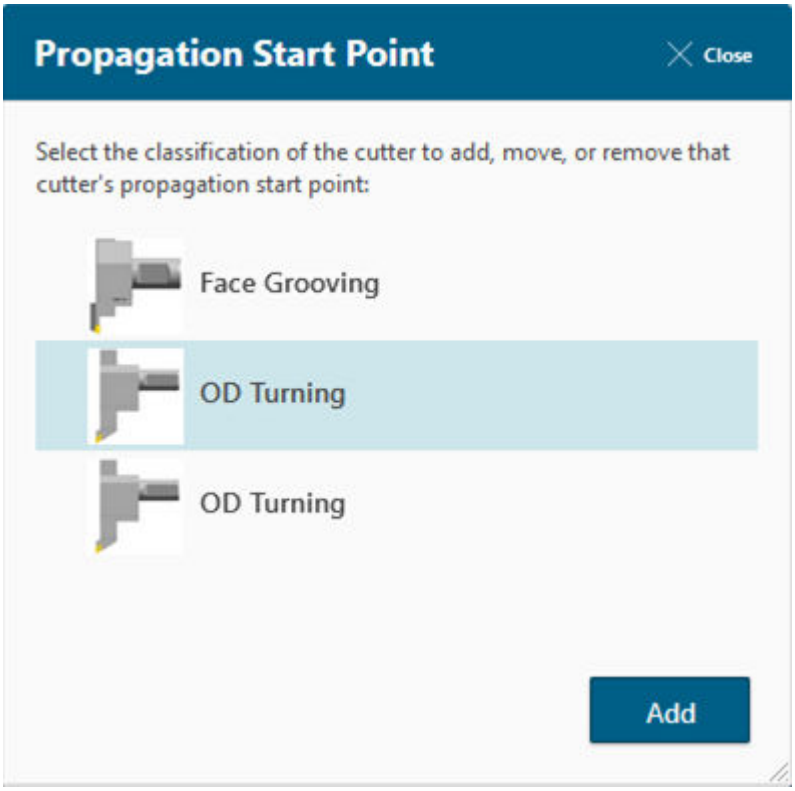

Note:

Depending on the cutter selected in the **Propagation Start Point** dialog box, you can click **Add**, **Move**, or **Remove** to take further action.

c. If needed, repeat this for each insert or cutter combination.

Deleting multitool cutters

You can remove a cutter from a multitool.

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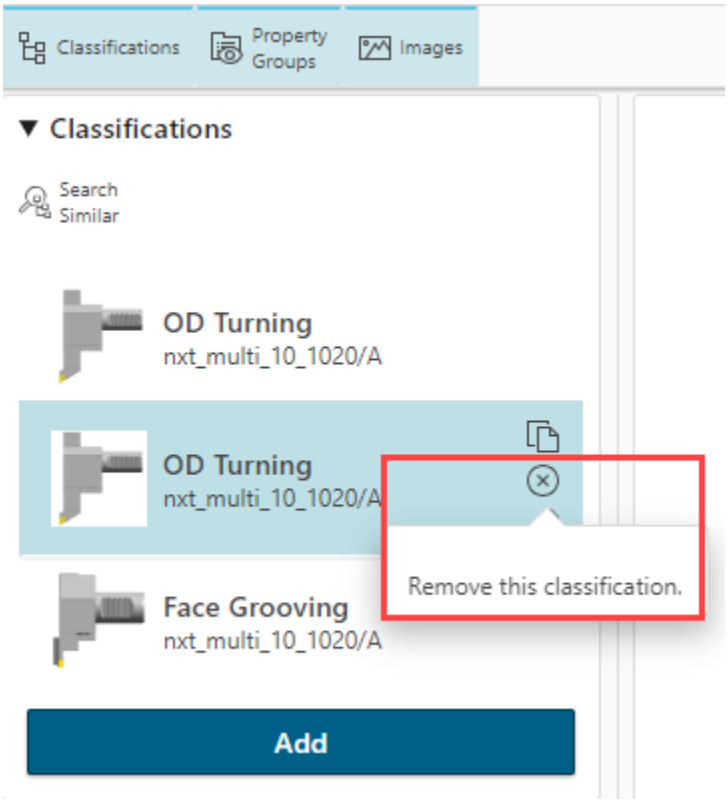

The following happens when you delete a multitool cutter classification object (ICO) using **Remove this classification** ⊗:

- Only the selected cutter ICO is deleted.
- The **Multitool [MT]** attribute in the **Multitool Data** group automatically updates the status of the cutters.

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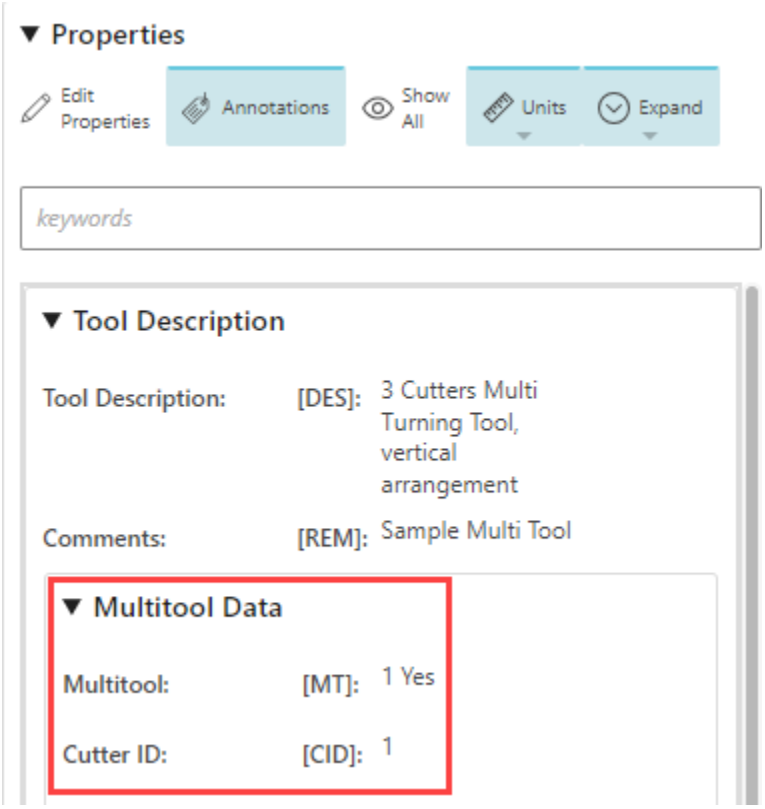
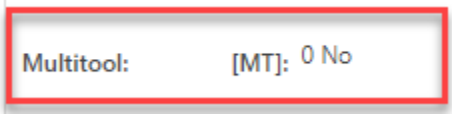


Note:

When you remove the second to last cutter, and only a single classification is assigned, the **Multitool [MT]** attribute status updates to **Multitool [MT]: 0 No** and changes the multitool into a tool.



- The system removes the related propagation start point (PSP) from the tool assembly.

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5. Using resources to manufacture parts in NX CAM

Getting started with creating tools in Resource Manager

Guidelines for creating tools

Use Resource Manager to build NC cutting tools that can be used in NX CAM.

In Active Workspace, you can use **Create NX Tool Assembly** to create or update an NX tool assembly part file, or to set the following features for use in NX CAM:

- **Identify cutting and non-cutting geometry**—Identifies the **CUTTING** or **NON_CUTTING** groups in an NX tool component part file. These are displayed in the machine tool builder and saved in the assembly part file in NX. For more information, see **Identify cutting and non-cutting geometry**.

- **Generate the spinning geometry**—Generates a spinning geometry. The information for this geometry comes from the **CUT** and **NOCUT** information of the tool components that are used in the tool assembly. For more information, see **Generate the spinning geometry**.

- **Set the tool junctions**—Adds the tooltip and tool mount junctions to the assembly part file. For more information, see **Set the tool junctions**.

- **Write NX part attributes**—Adds specified tool mount and tooltip junctions into the part properties in the assembly part file. Also sets the **CAM_TOOL_ATT_X**, **CAM_TOOL_ATT_Y**, and **CAM_TOOL_ATT_Z** properties. For more information, see **Write NX part attributes**.

Identifying cutting and non-cutting geometry

Use to identify cutting and non-cutting geometries, or to assign cutting geometry colors, in a tool component part file.

NX identifies the tool component cutting and non-cutting geometries, as they are defined in the NX tool component part file, using one of the following attributes, in the following order: **Body**, **Color**, and **Feature Group**.

> Tip:
>
> It is recommended that you assign only a single activity, such as a cutting activity only, a non-cutting activity only, or a combination of both, to the attributes in a tool component part file.

Opening the tool component part file in NX, you can review the following tool component attributes to make sure they are properly prepared for later automatic processes:

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- **Body**

  When you name a body of the tool component part file **CUT**, the body component is identified as the cutting geometry. The remaining components are identified as non-cutting geometry.

- **Color**

  Items with the color **Light Gray** are identified as the cutting geometry. All other colors are identified as non-cutting geometry.

  Specific colors can be assigned to the cutting geometry in the tool component part file.

  - Colors that are defined in the DIN Standard

    ■ Color for **CUT**: 80% Gray (**Light Gray**) ± 1 %

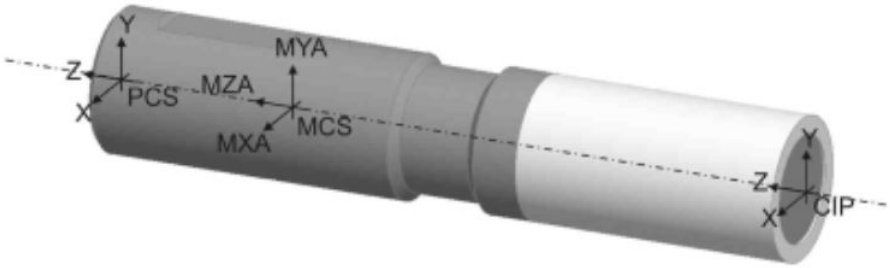
    Valid **Red**/**Green**/**Blue** (RGB) values: **Red**: 204 **Green**: 204 **Blue**: 204

    > Note:
    >
    > The RGB values must be between 202 and 206.

    ■ Color for **NOCUT**: 50% Gray (**Medium Gray**) ± 1 %

    Valid **Red**/**Green**/**Blue** (RGB) values: **Red**: 125 **Green**: 125 **Blue**: 125

    > Note:
    >
    > The RGB values must be between 125 and 129.



  - Colors in NX

    The following are colors used in the default NX color palette and are within the tolerance range that is defined in the DIN standard:

    ■ Color for **CUT**: **Light Gray**

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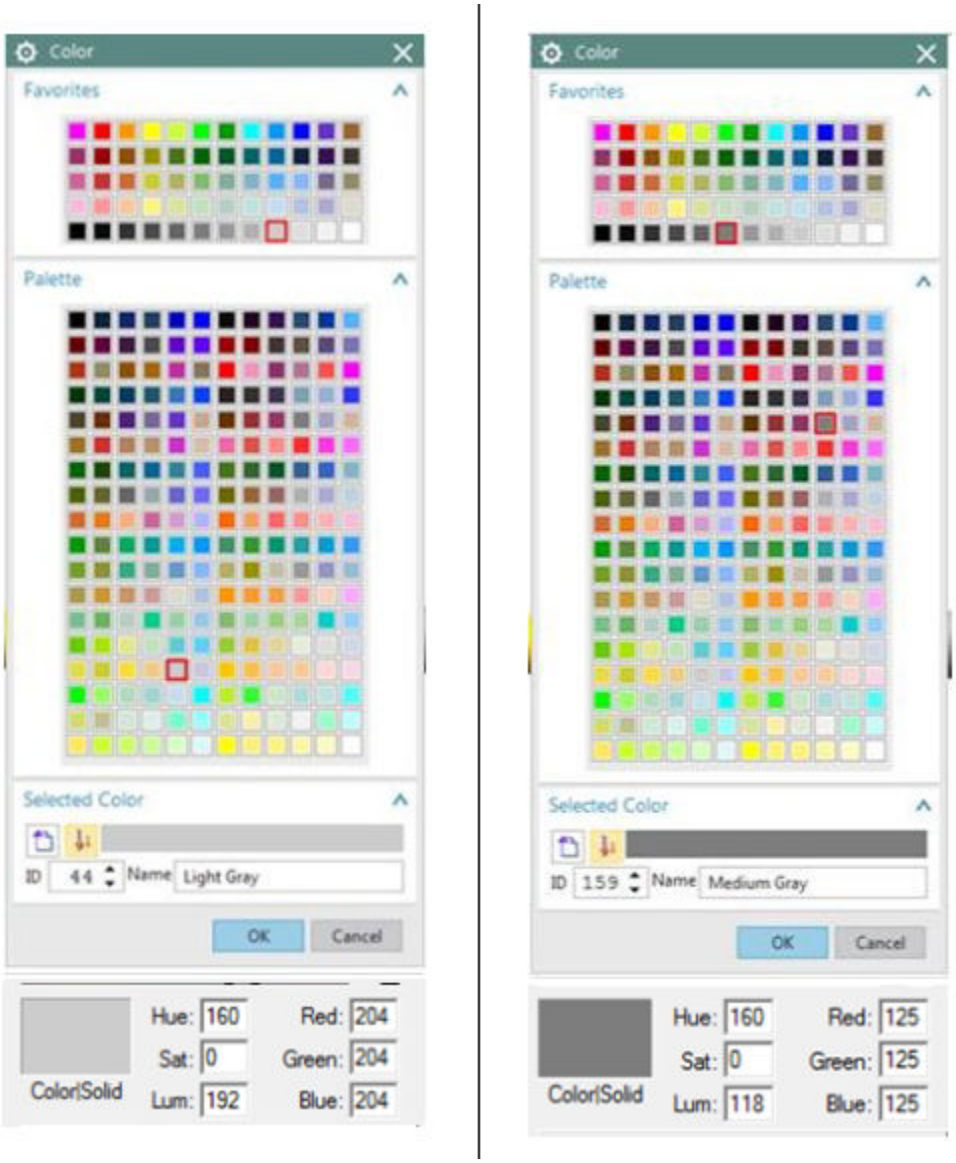
Valid **Red**/**Green**/**Blue** (RGB) values: **Red**: 204 **Green**: 204 **Blue**: 204

- Color for **NOCUT**: **Medium Gray**

  Valid **Red**/**Green**/**Blue** (RGB) values: **Red**: 125 **Green**: 125 **Blue**: 125

> Note:
>
> Depending on the color palette being used, some colors may have a different appearance, such as when NX imports a STEP file, a similar matching color from the NX color palette is used.



- **Feature Group**

  The two feature groups include the following:

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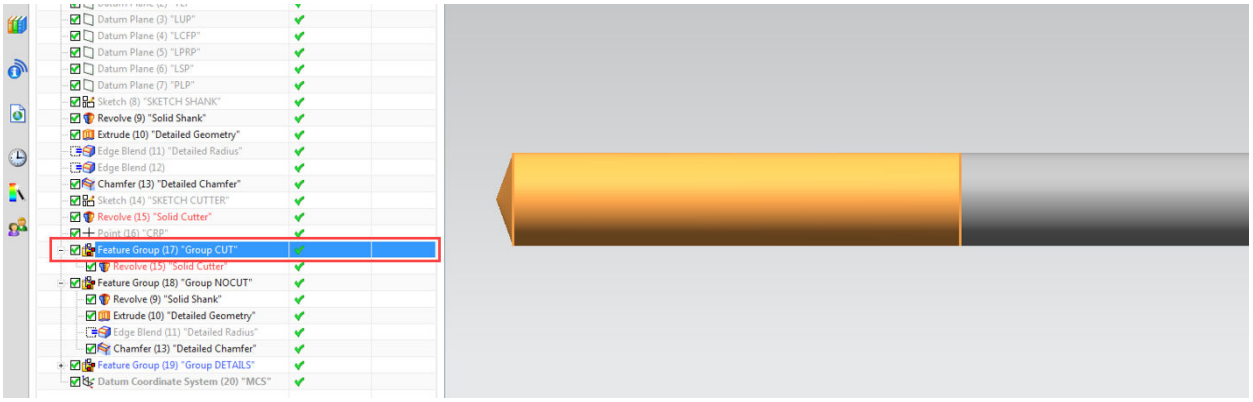
- The feature group **CUT** defines the cutting geometry.

  NX identifies a cutting feature group when a space character is added before the word **CUT** in the name of the cutting feature group, such as **add feature group name**-space character-**CUT**.

- The remaining components are identified as non-cutting geometry.

> Note:
>
> Below is an example of a component, such as a drill, that includes cutting and non-cutting geometries.



- **Validation**

  Using the **Machine Tool Builder** you can verify that the cutting and non-cutting geometries are identified properly for the tool assembly.

> Tip:
>
> Selecting the **CUTTING** and **NON_CUTTING** attributes highlights the corresponding geometry, as in the following examples:

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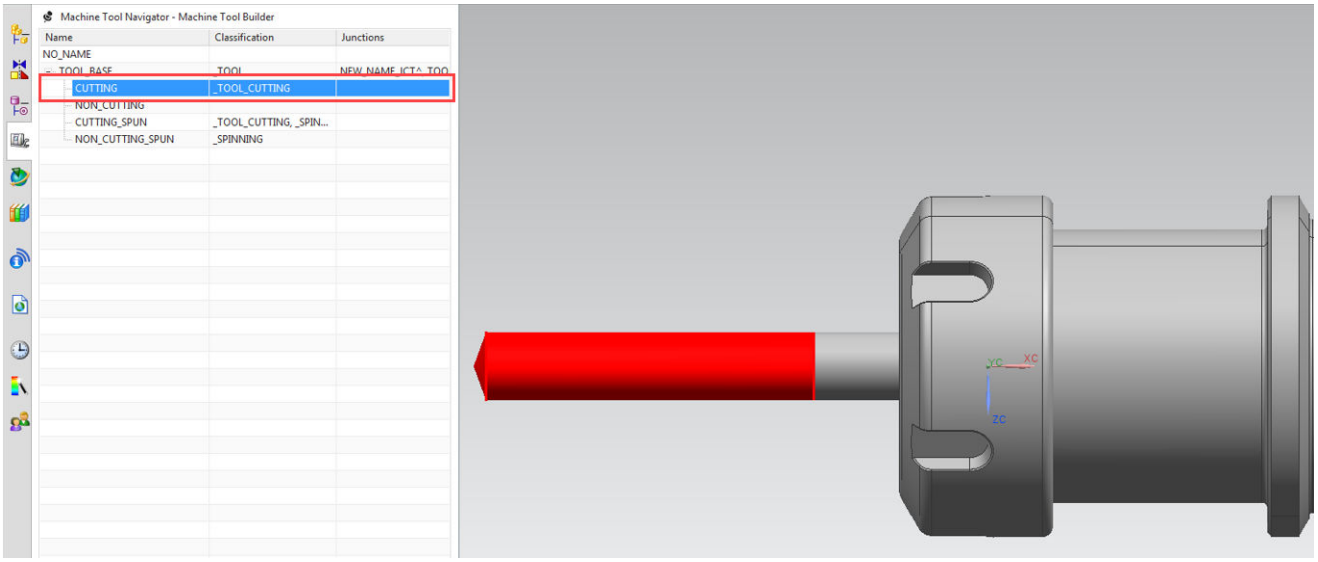

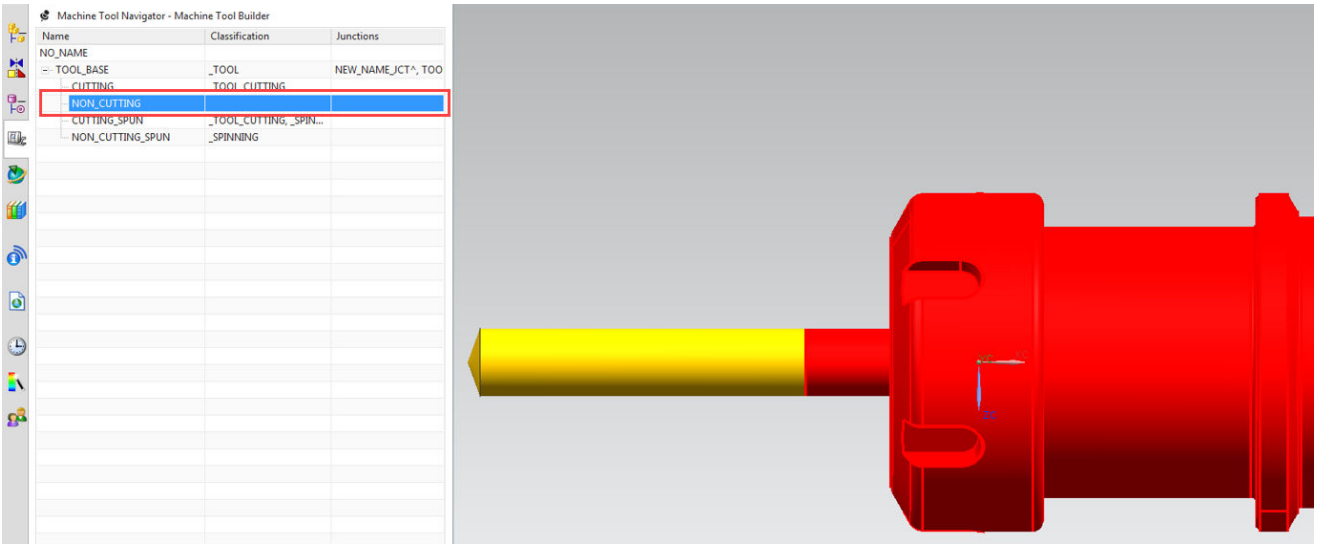

Generating spinning geometry

By selecting cutting and non-cutting geometries, you can generate a spinning geometry.

> Note:
>
> You must define the cutting and non-cutting geometries in **Identify cutting and non-cutting geometry** before generating the spinning geometry.
>
> This function is not available for turning tools.

Using the **CUTTING_SPUN** and **NON_CUTTING_SPUN** settings in the **Machine Tool Builder** section, you can confirm the cutting and non-cutting spinning geometry of a tool assembly.

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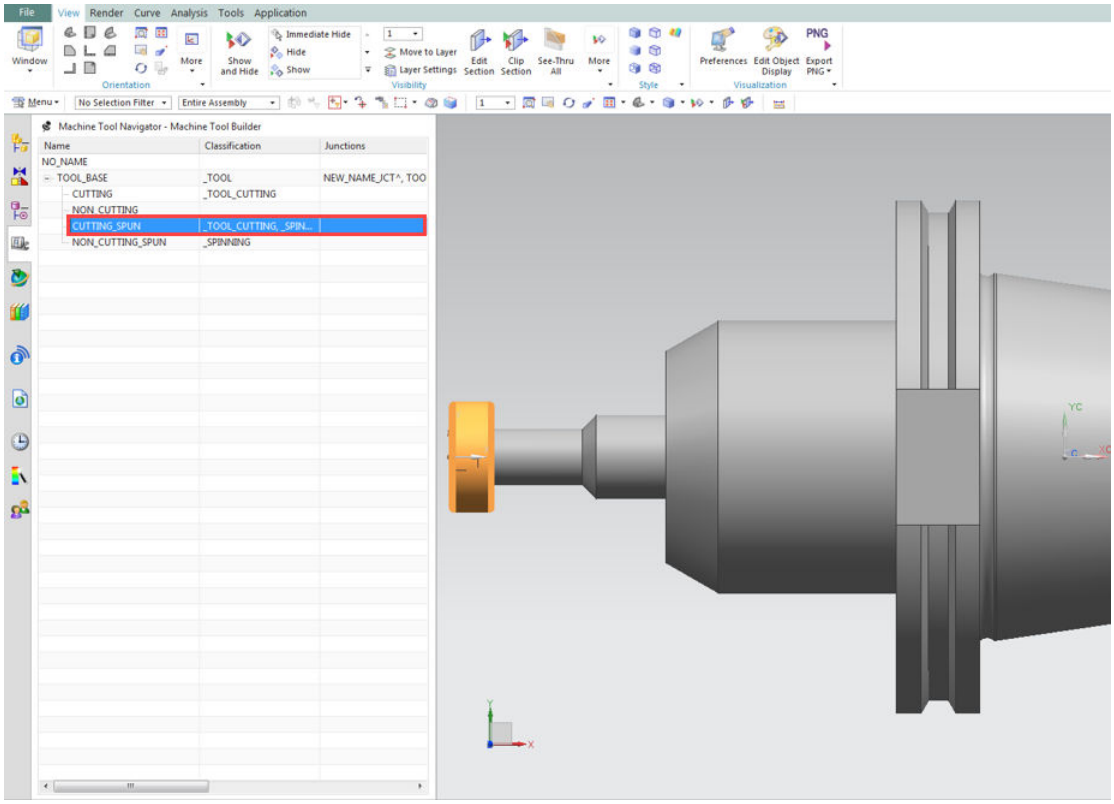
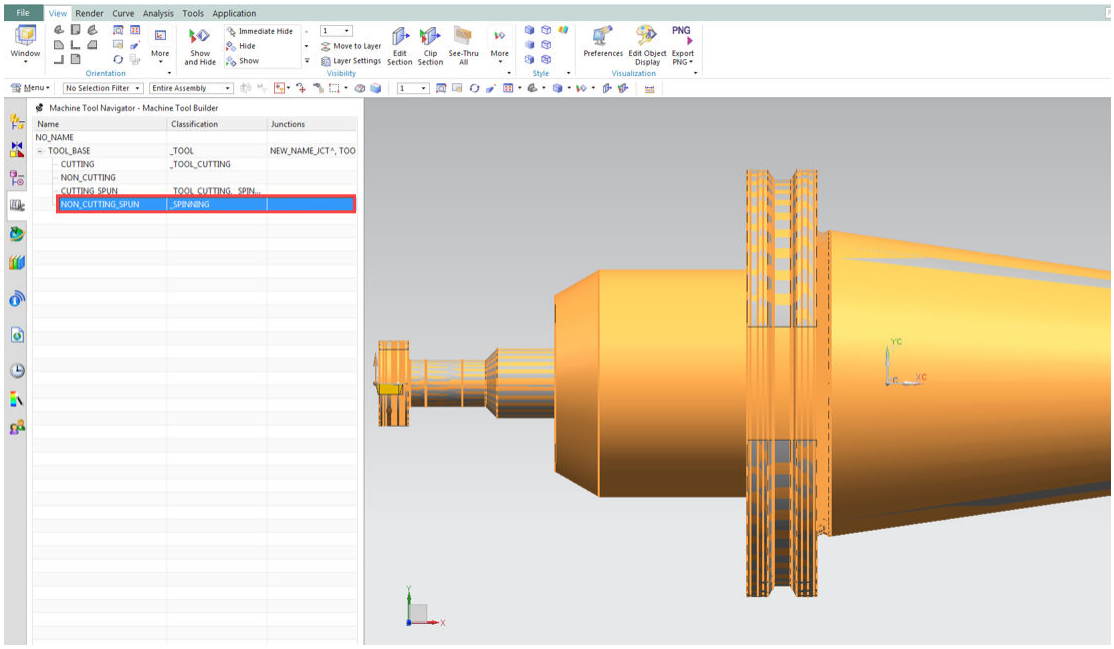
Selecting the **CUTTING_SPUN** or **NON_CUTTING_SPUN** setting highlights the corresponding geometry.





Setting tool junctions

You can use set tool junctions to properly build a tool assembly.

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- **Logical structure and tool holder**

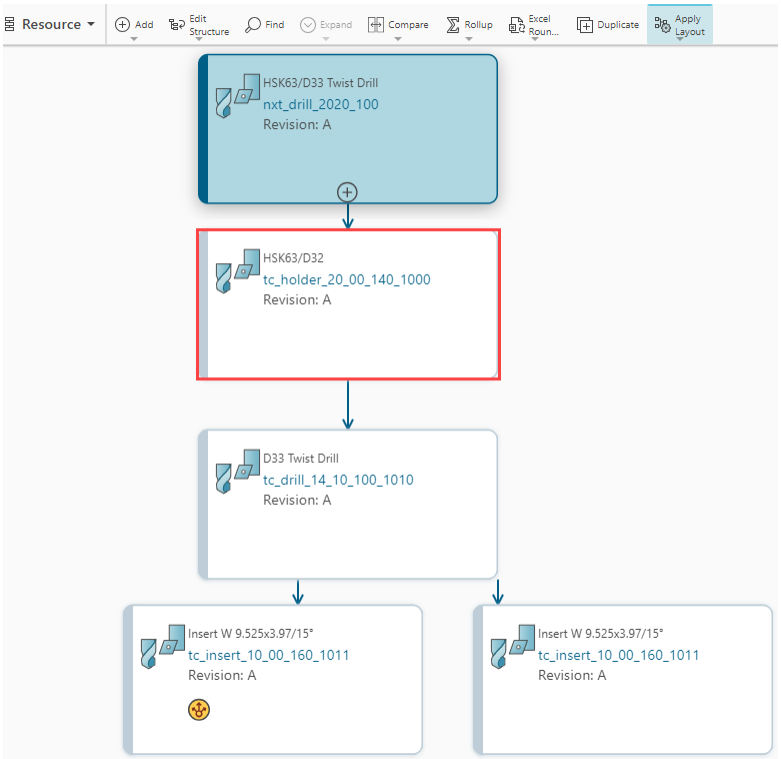
  Tool holder components are used by the **Auto Assembly** functionality to find the **TOOL_MOUNT_JUNCTION**.

  If the tool holder is not properly defined, the **TOOL_MOUNT_JUNCTION** cannot be found.

  A tool holder structure must include the following:

  - Only a single tool holder can be positioned directly below the tool assembly root node. Other drill or insert components must be positioned below the tool holder.
  - Below are three different examples of tool assemblies that have the tool holder properly defined:

    ■ **Drilling tool**



    ■ **Milling tool (with inserts)**

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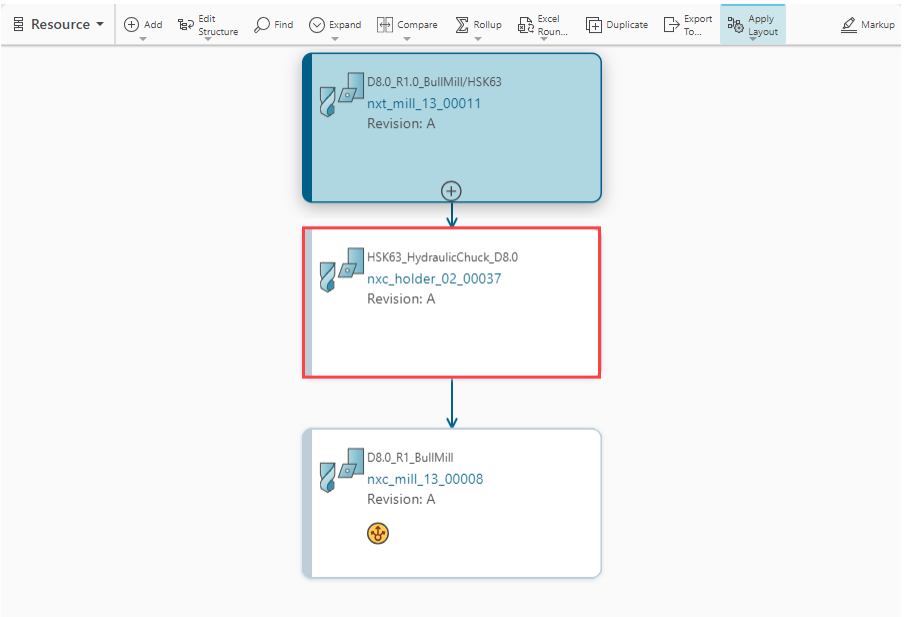
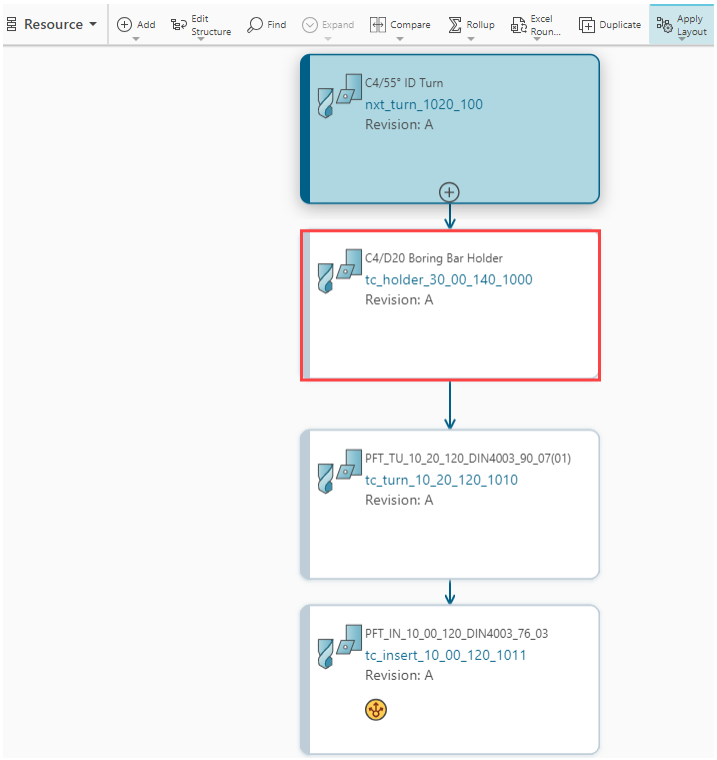


■ **Turning tool**



- **Turning: tracking point**

  The tracking point for turning tools must be defined for the tool tip junction. A tool tip junction tracking point that is not defined in Teamcenter cannot be evaluated by the **Create/Update NX Tool Assembly Part File** functionality.

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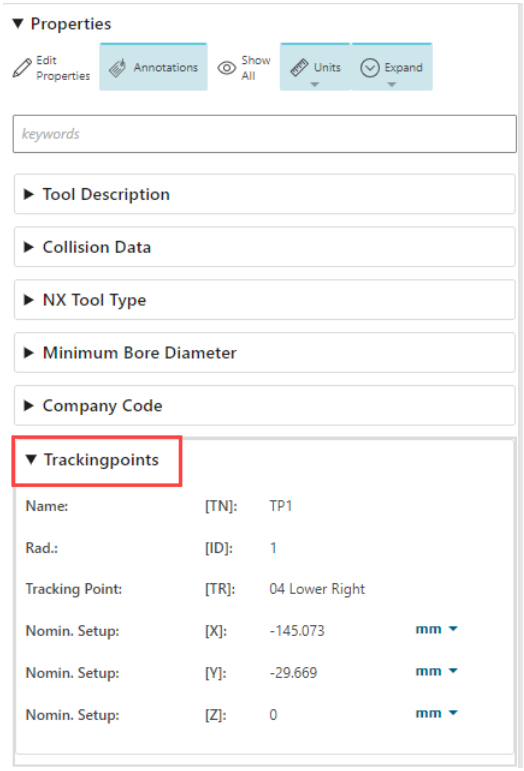
Note:

The **09 Middle** value is not supported by the functionality. Instead, it is used as a theoretical point that does not make sense for the tool path calculation, because this point cannot be measured by a presetter.

Terms, such as **04 Lower Right** and **02 Upper Left**, as defined by the tool's tool mount junction, are always used for the X/Y-plane.



The tool tip junction is positioned at the same orientation as the tool mount junction. When the tool is displayed in NX, the X-axis points to the right and the Y-axis points to the top. The cutting geometry, left, right, upper, and lower tracking point values must be accurate.

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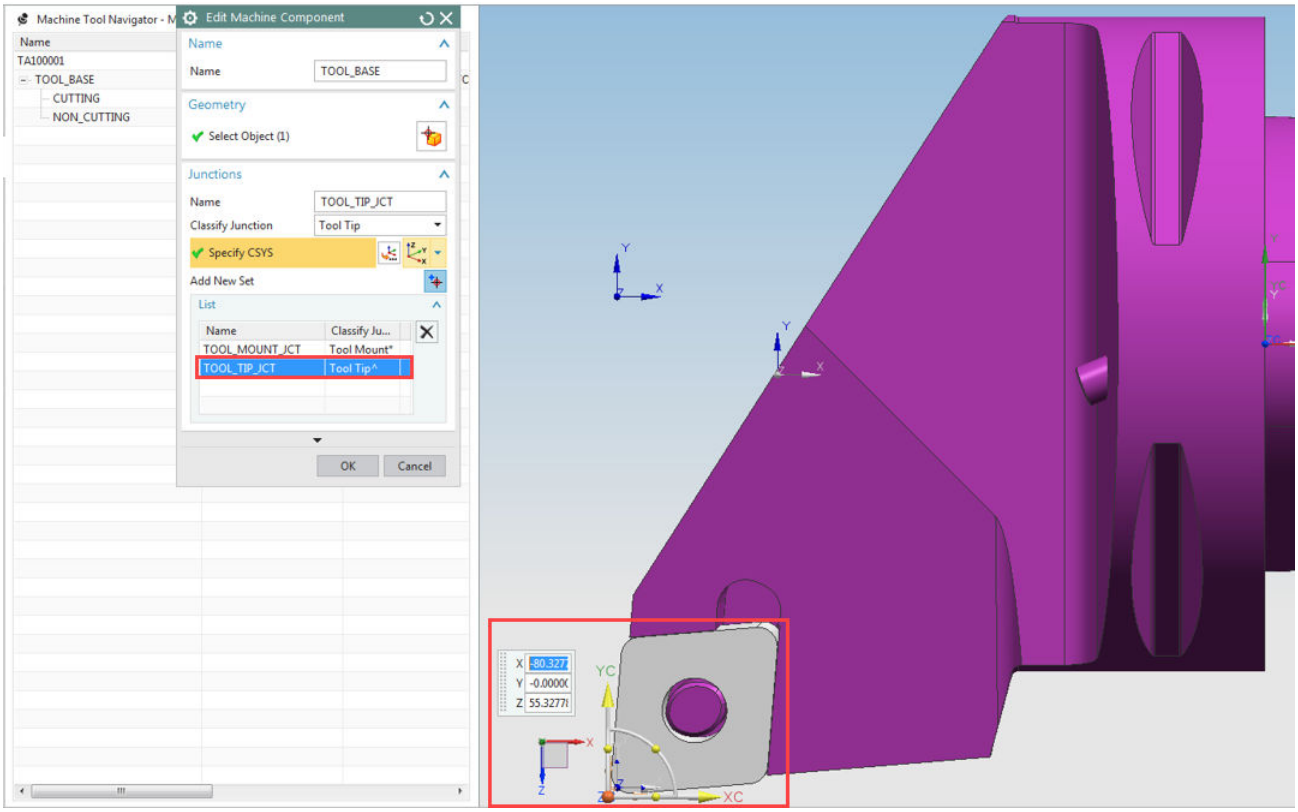

- **Tool mount junction**

  A tool mount junction is defined in the following ways:

  - **SIEMENS_TOOL_MOUNT**

    A tool mount junction is always defined by one coordinate system in the tool's tool holder. For more information see **Logical structure and tool holder**.

    When a tool holder includes a **SIEMENS_TOOL_MOUNT**, this is used as the tool's tool mount junction.

  - **MCS**

    When a tool holder does not include a **SIEMENS_TOOL_MOUNT** coordinate system, the tool holder's **MCS** coordinate system is used for the tool mount junction.

    In NX CAM, the tool's cutting point is on the negative X-axis. Tool components based on DIN4003, or tools downloaded from tool vendor's web pages, have a different orientation and the cutting point is on the positive Z-axis. The **Create/Update NX Tool Assembly Part File** functionality automatically transforms the orientation of the tool component's coordinate system to the orientation required in NX CAM.

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The tool mount junction cannot be evaluated when a tool holder does not include a **SIEMENS_TOOL_MOUNT** or a **MCS** coordinate system.

The **SIEMENS_TOOL_MOUNT** coordinate system is not mandatory, and the algorithm also works with components that only have **MCS** defined.

- **NX tool tip junction**

The evaluation of the tool tip junction is different for turning tools and drilling and milling tools.

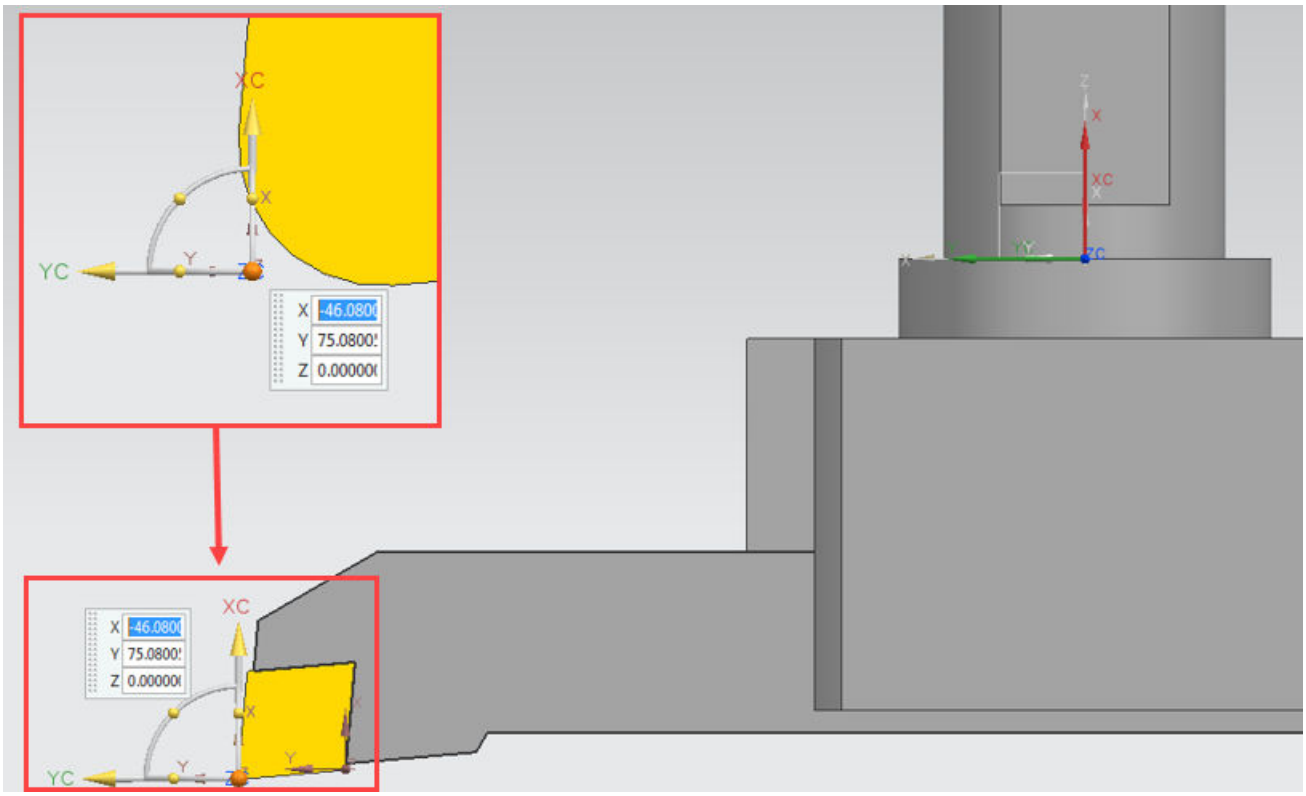
- **Turning tools**

The tracking point defines the direction from which the tool tip is searched. For more information see Turning: tracking point.

The presetter algorithm identifies the first point where the cutting component is touched from the specified direction. This point is used as the tool tip junction.

> Note:
>
> The algorithm always evaluates the tool tip junction based on the 3D geometry.

■ The **03 Lower Left** tool tip is magnified and highlighted.



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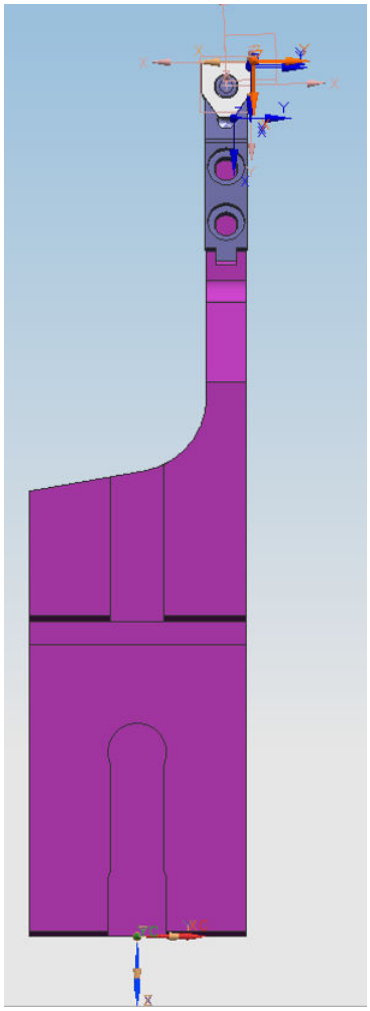
For turning tools, the tool tip and the tool mount junctions must be assigned the same Z-level.

> Note:
>
> The algorithm automatically transforms the tool tip junction to the Z-level of the tool mount junction.

- The tool tip is highlighted.



- The tool tip is not directly on the cutting insert, but is on the same Z-level as the tool mount.

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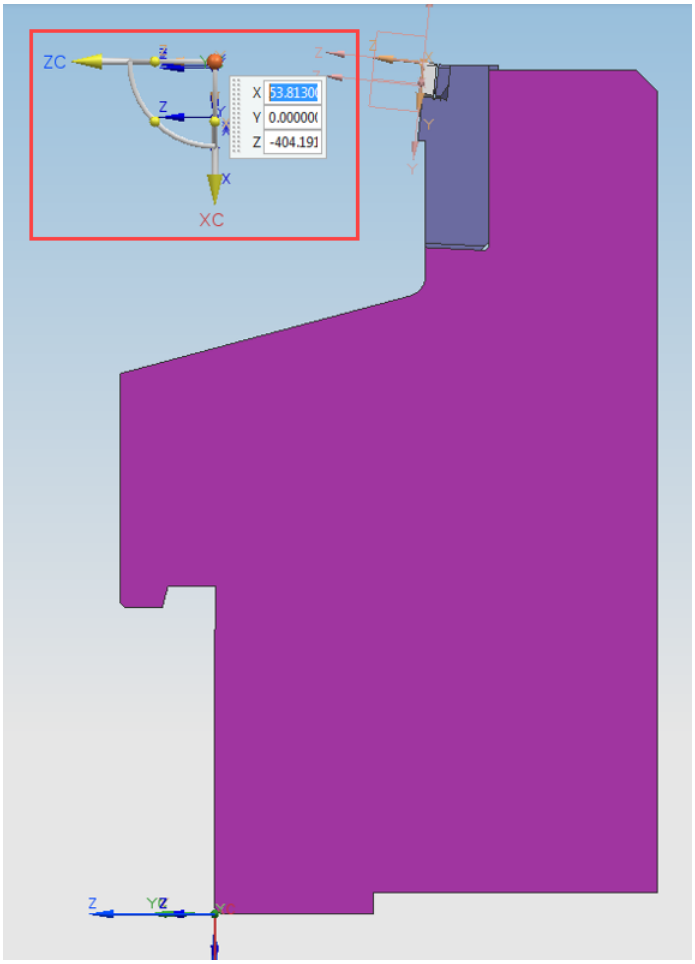

- The tool mount is highlighted.

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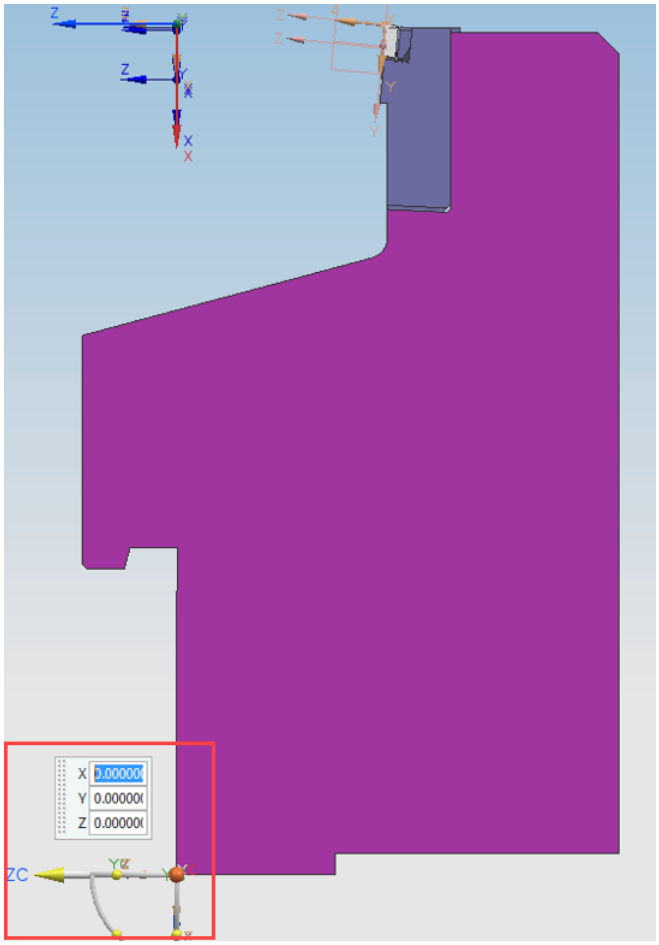

- **Drilling and milling tools**

  When a CIP coordinate system is defined in the tool's cutter component, it is used as the tool tip junction.

  When a CIP coordinate system is not defined, the algorithm evaluates the tool tip junction based on the 3D geometry.

  > Note:
  >
  > The point that is the furthest in the X-axis direction is used as the tool tip junction.
  >
  > The CIP coordinate system is not mandatory. The algorithm also works with components without the CIP coordinate system.

  For all tool types, the orientation of the tool tip junction is the same as the tool mount junction.

- **Validation**

  You can validate the position and orientation of the tool tip and tool mount junctions in the NX Machine Tool Builder by using the **Edit Machine Component** dialog box.

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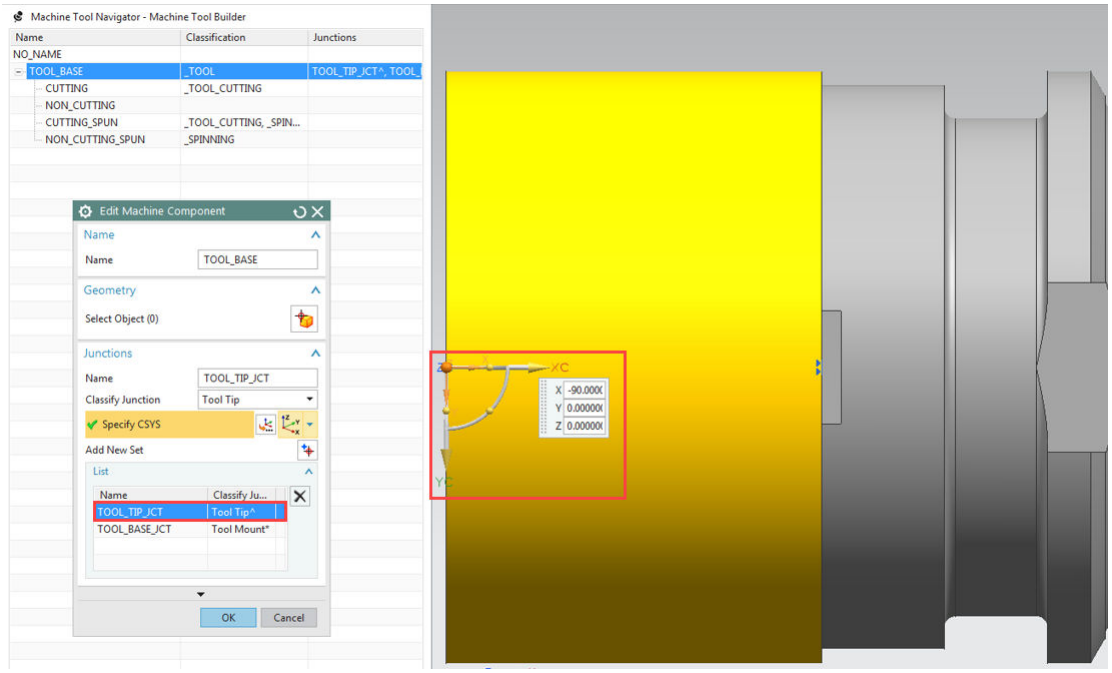
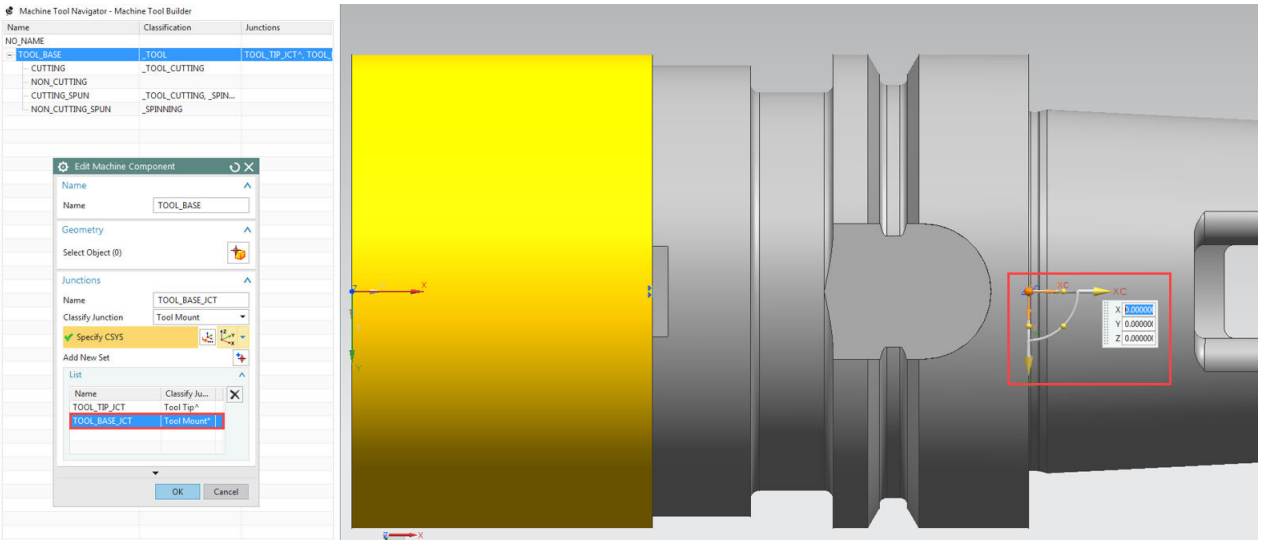
Selecting **Tool Tip** highlights the tool tip junction.



The **Tool Mount** is highlighted when the tool mount junction is selected.



Writing NX part attributes

You can use the **Update NX Tool Assembly Part File** menu command to select various part attributes.

When you execute NX Integration functions in Teamcenter, by default the part attributes are not written into the part file.

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> Note:
>
> The **Set tool junctions** setting in the **Update NX Tool Assembly Part File** menu command must be selected to use this function.

Attributes must be mapped before you synchronize **Setup X**, **Setup Y**, and **Setup Z** attribute values from NX to Teamcenter.

For more information see Synchronize attribute values from NX to Teamcenter.

> Note:
>
> When you reload the resource NX Integration functions in Teamcenter, the updated part file attribute values are displayed.
>
> The setup values are immediately updated in Teamcenter when the **Auto Assembly** functionality is used.

Teamcenter and NX Validation

- **Validation in Teamcenter**

  The **Nomin. Setup X** and **Nomin. Setup Y** settings are empty and need to be defined before you create a new **Update NX Tool Assembly Part File** in Teamcenter.

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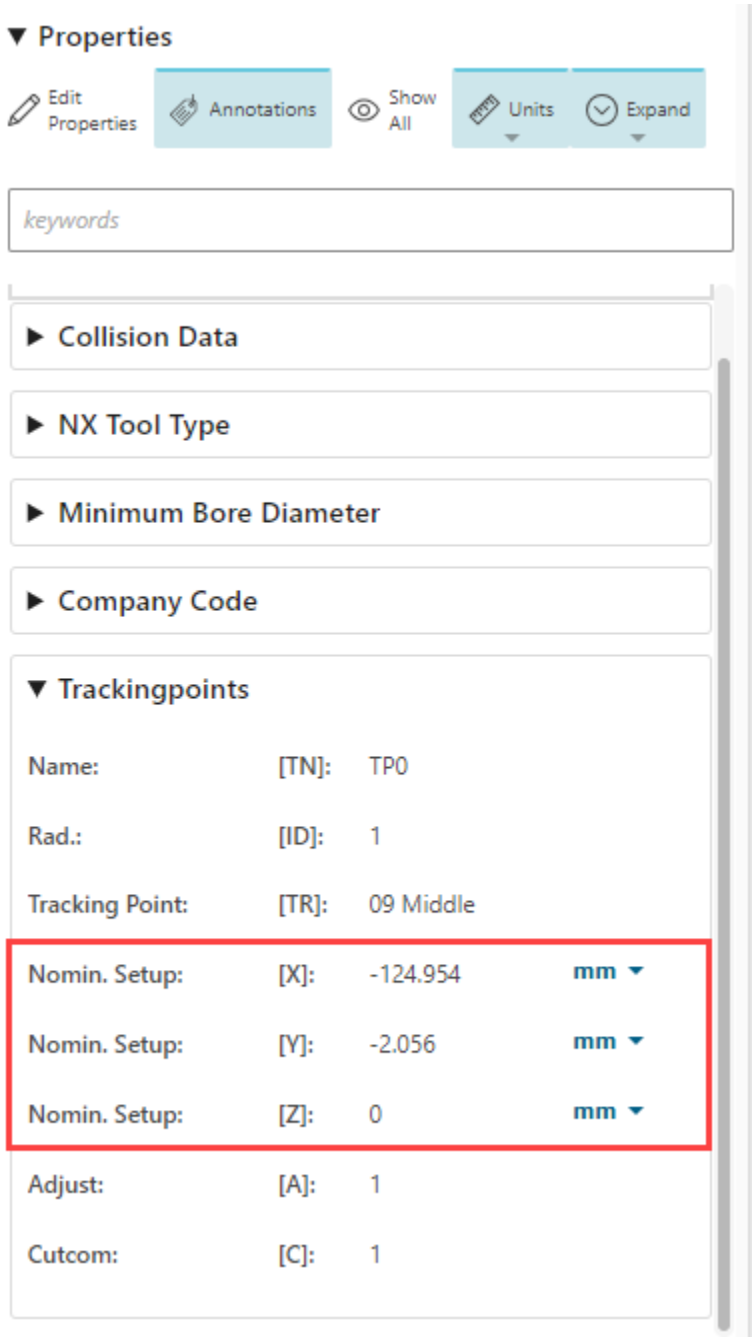

- **Validation in NX**

  When you create a new **Create/Update NX Tool Assembly Part File**, and open the part file in NX, the **CAM_TOOL_ATT_X**, **CAM_TOOL_ATT_Y**, and **CAM_TOOL_ATT_Z** part attributes are displayed with the proper values.

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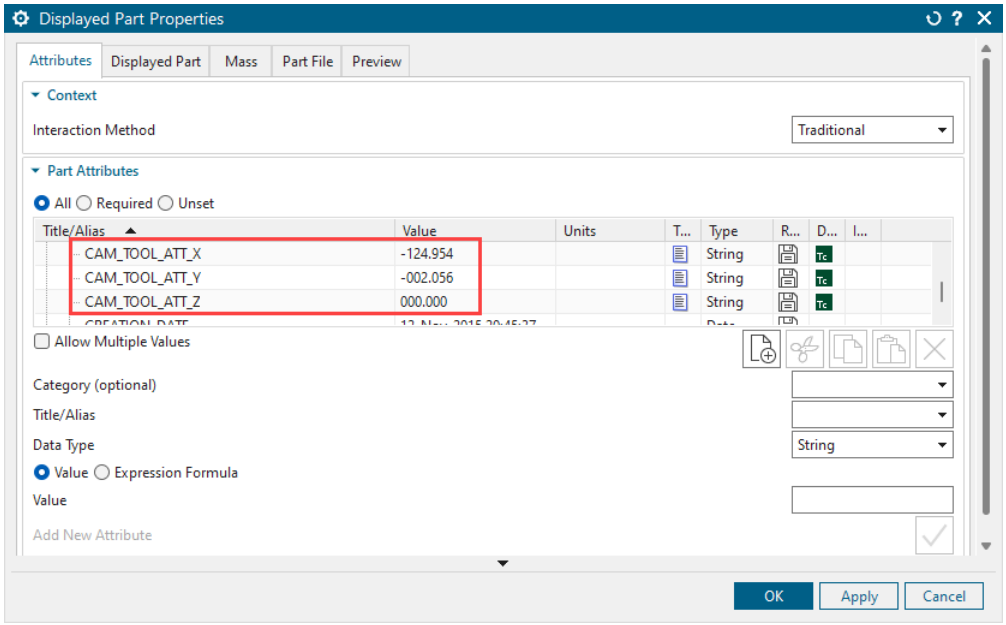

Note:

The **Graphics Builder** must be configured before you use the **Create/Update NX Tool Assembly Part File**.

Create NX tool assembly

After you create a tool assembly, Active Workspace helps you create or update the NX tool assembly part file. To do this, you capture CAM-related tool information required by the machine tool builder in NX, so the tool assembly can be used for tool path simulation.

Note:

Before you create the NX tool assembly, confirm that the graphics builder is correctly installed and configured.

1. Select the root node of the tool assembly.

2. Click **Manage** and choose **Create NX Tool Assembly**.

   Active Workspace displays the **Create NX Tool Assembly** dialog box.

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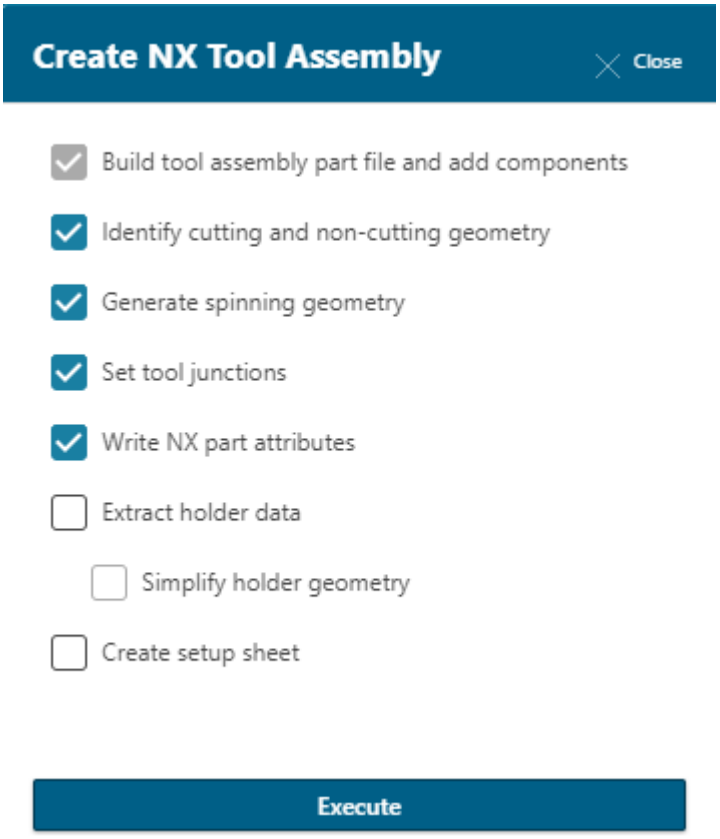

3. Select options in the dialog box.

> Note:
>
> The first option in the **Create NX Tool Assembly** dialog box is the focus of this procedure and is required. You can also select the options **Extract Holder Data** and **Create Setup Sheet**.

Option	Description
Build tool assembly part file and add components	Builds NX tool assembly part file and adds components to the assembly structure. These steps are mandatory when you create the NX tool assembly. This option is selected by default and cannot be unselected.
Identify cutting and non-cutting geometry	The solids in the tool components are added to the **CUTTING** or **NON_CUTTING** groups in NX. These are displayed in the machine tool builder and saved in the assembly part file in NX.
Generate spinning geometry	The **CUTTING_SPUN** and **NON_CUTTING_SPUN** geometry are saved with the tool assembly for milling and drilling tools. The information for this geometry comes from the **CUT** and **NOCUT** information of the tool components that are used in the tool assembly.

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Option	Description
	This option is not available for turning tools.
Set tool junctions	The tooltip and tool mount junctions are added to the assembly part file.
Write NX part attributes	If the tool mount and tooltip junctions are defined in the assembly part file, NX calculates the setup offsets based on these two junctions, and writes them into the part properties in the part file, setting the **CAM_TOOL_ATT_X**, **CAM_TOOL_ATT_Y**, and **CAM_TOOL_ATT_Z** properties. Additionally, NX system tracking points are imported into the resource for milling and drilling tool assembly classes.
Extract Holder Data	Tool holder parameters that are required by NX to calculate collision avoidance are automatically imported into Teamcenter.
Simplify holder geometry	Outputs fewer holder segments, using a tolerance and ignoring undercut holder geometry.
Create Setup Sheet	Generates setup sheets, which are based on templates that are saved with the class.

4. Click **Execute**.

   An NX assembly part file is created in which all tool components are correctly positioned. The part file includes the additional tooling information. The part properties are mapped to Teamcenter. For example:

   CAM_TOOL_ATT_X is mapped to the resource library attribute **-45002 Nomin. Setup X**.

   CAM_TOOL_ATT_Y is mapped to the resource library attribute **-45003 Nomin. Setup Y**.

   CAM_TOOL_ATT_Z is mapped to the resource library attribute **-45002 Nomin. Setup Z** (for turning tools).

   Note:

   The NX part attribute to Classification mapping must be configured.

   Your tooling administrator must enable the synchronization of **Setup X**, **Setup Y**, and **Setup Z** attribute values.

   Refer to the topic Synchronize attribute values from NX to Teamcenter for a list of more attributes that get mapped.

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Obtain data on NX CAM tool assembly

Three options available when you click the **Manage** icon provide information for the associated NX CAM tooling assembly.

Extract holder data

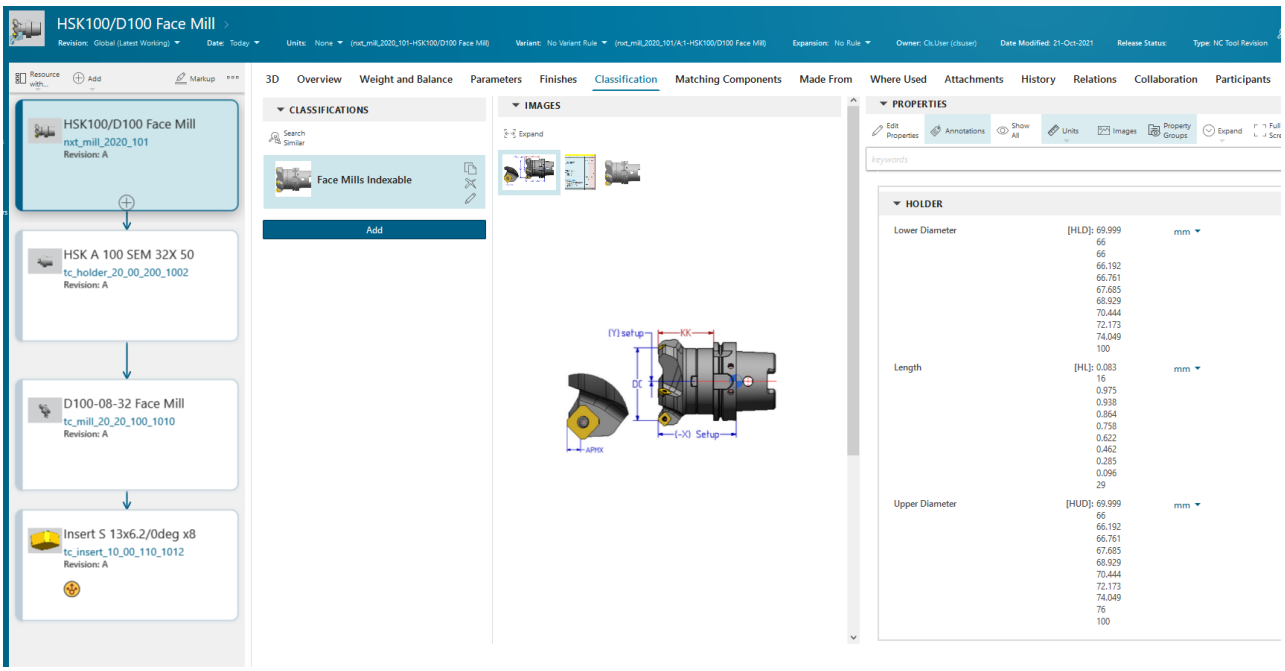
1. Select any node in the tool assembly.

2. Select **More commands** •••→**Manage** →**Extract Holder Data** .

   If the holder data is successfully extracted, you will see the message: **Holder data was successfully extracted**.

   The holder data is added as classification data for the holder.

> Note:
>
> The available tabs and their order can differ depending on what templates were imported and where you are in Active Workspace.



> Note:
>
> The **Extract Holder Data** command always creates data for a simplified holder.

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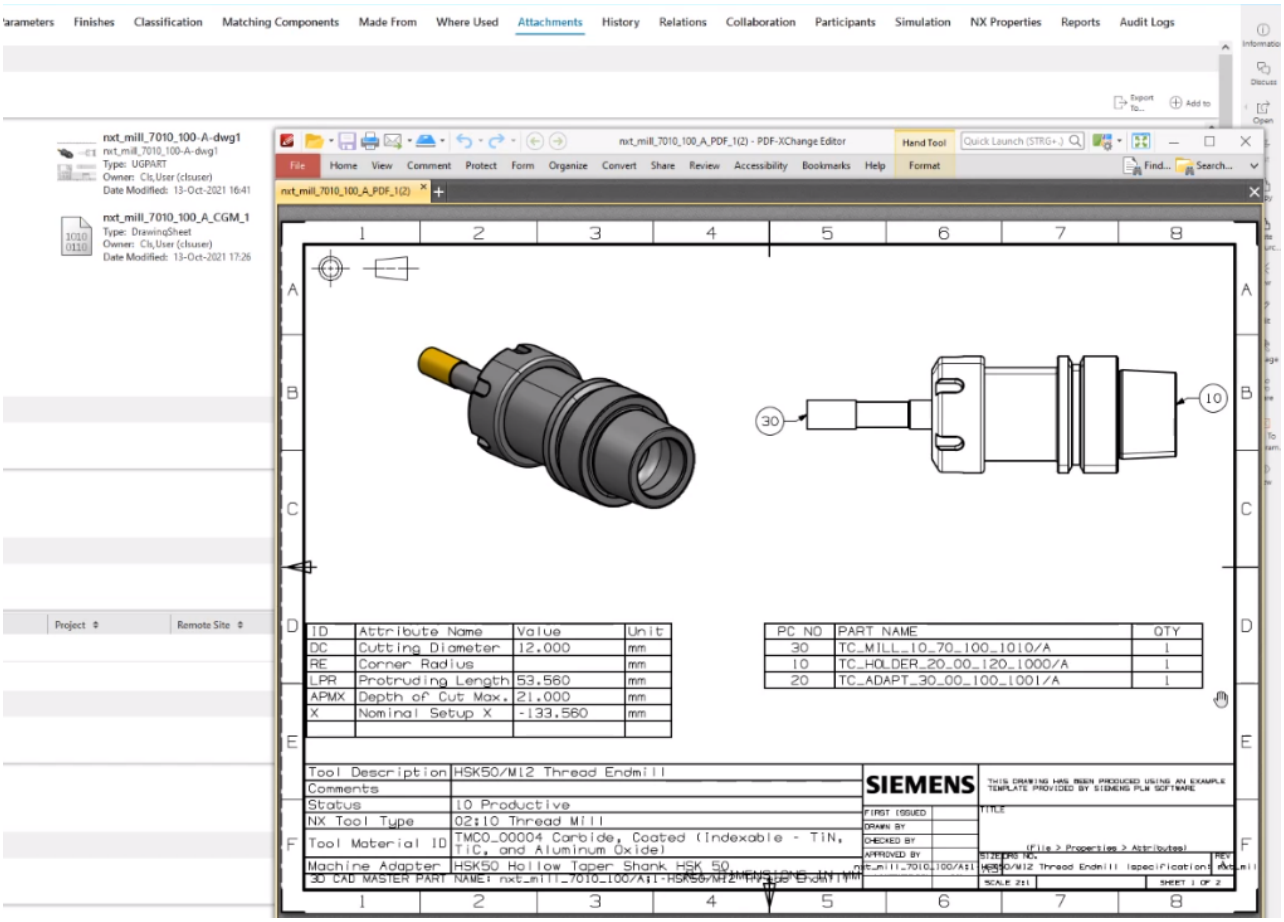
Create Setup Sheets

1. Select any node in the tool assembly.
2. Select **More commands** •••→**Manage** →**Create Setup Sheet** .

   The setup sheet is available from the **Attachments** tab.



Determine ability to retrieve tool assembly in NX CAM

1. Select any node in the tool assembly.
2. Select **More commands** •••→**Manage** →**Check NX CAM Tool Retrieve** .

   You see a message indicating whether the tool can be successfully retrieved in NX CAM:

```
==== Report for tool
"nxt_mill_7010_100/A-HSK50/M12 Thread Endmill" ====
  ---- Results from "Check NX CAM tool retrieve":
  Tool can be opened in NX-CAM without any errors.
```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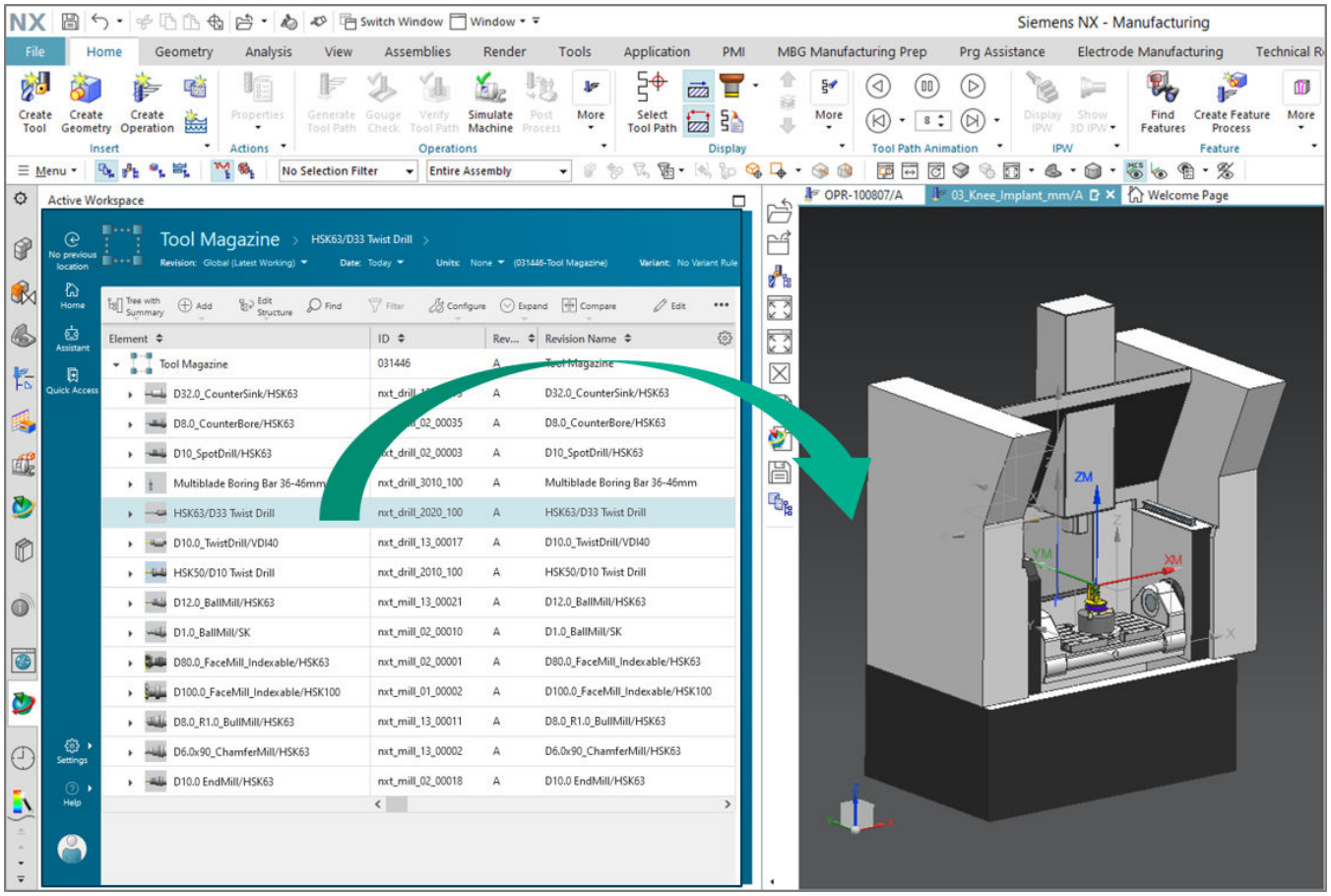
Adding resource items to NX CAM

Using Active Workspace, you can add a Manufacturing Resource Library (MRL) object to NX CAM.

You add resources and devices to an existing NX CAM assembly by selecting the object in MRL and dragging it to NX CAM.



You can select one or more MRL resources to add to an NX assembly. The following tool structure or **Part Manufacturing** object types that can be selected and dragged to NX include:

- (1) **Resource**, such as a **Fixture** or **Device**
- (2) **Tool**
- (3) **Machine**

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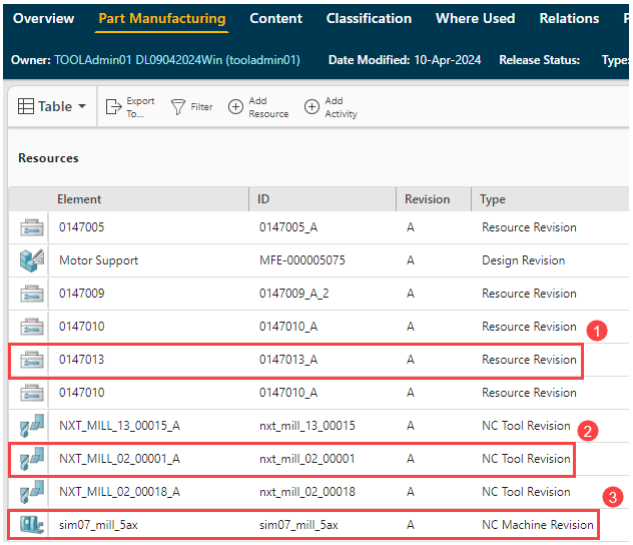
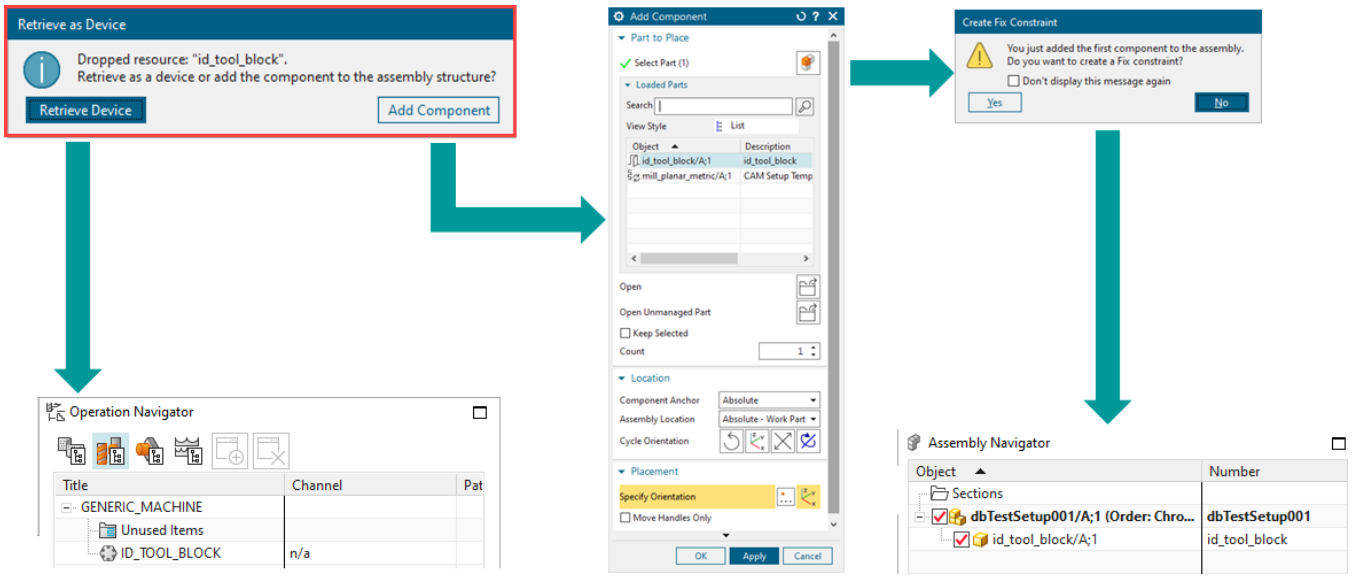


When you select a resource that is not a tool or a machine and drag it to an opened NX assembly, the **Retrieve as Device** dialog box is displayed.

Retrieve Device adds the resource as a device to the NX CAM **Operation Navigation** tree.

Add Component adds the resource as a component to the NX CAM **Assembly Navigation** tree.

> Note:
>
> When two or more devices are added to NX, the **Retrieve as Device** dialog box is displayed for each object.



MRL resources can be added to the NX CAM assembly from the following MRL locations:

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- **Manufacturing Resource Library**.
- **Folders**.
- Global search.
- **MRL Saved Searches**.
- **Home**.
- **Revisions** when opened in Active Workspace.
- **Part Manufacturing**, for **NC Setups** when opened in Active Workspace, such as **NC Machining Operation** Item Revisions.

Managing graphics for use in NX

Graphics for use in NX

You can create, update, or prepare graphics in Active Workspace for use in NX.

Use Active Workspace to prepare graphics for use in NX

You can use **Prepare Graphics for NX** in Active Workspace to prepare an object, using a graphics template, to generate a 3D model. This object can then be used in NX to create graphics using classification attribute values. For more information, see **Prepare a graphic for use in NX**.

> Note:
>
> After preparing the graphic in Active Workspace, you must then use NX to complete the graphic creation process. For more information, see Managing tool assemblies between Manufacturing Resource Library and NX.
>
> If you use **Prepare Graphics for NX**, you don't need to use the graphics builder.

Use a template to create or update graphics in NX

You can use **Update Graphics** in Active Workspace to create or update the graphics of an object using its classification properties. For more information, see Understanding how you can create graphics for a class based on templates or Create graphics for a class based on templates.

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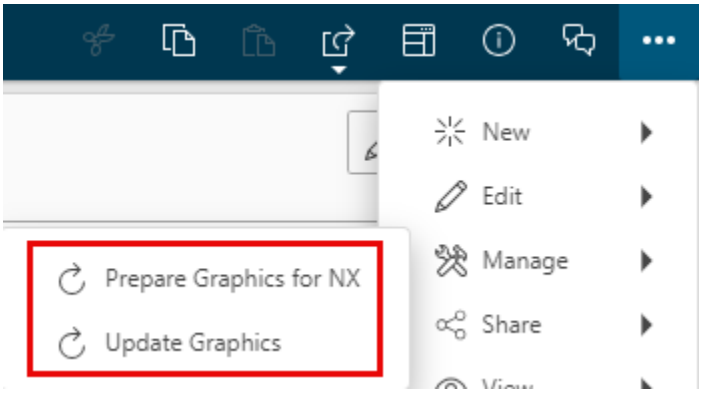
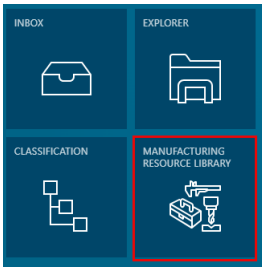


Prepare a graphic for use in NX

In Active Workspace, you can prepare graphics for use in NX without having to use the graphics builder.

Procedure

1. In Active Workspace, click the **Manufacturing Resource Library** tile.



2. Select or open a resource in the Manufacturing Resource Library (MRL) workspace.

3. Select the **Classification** tab.

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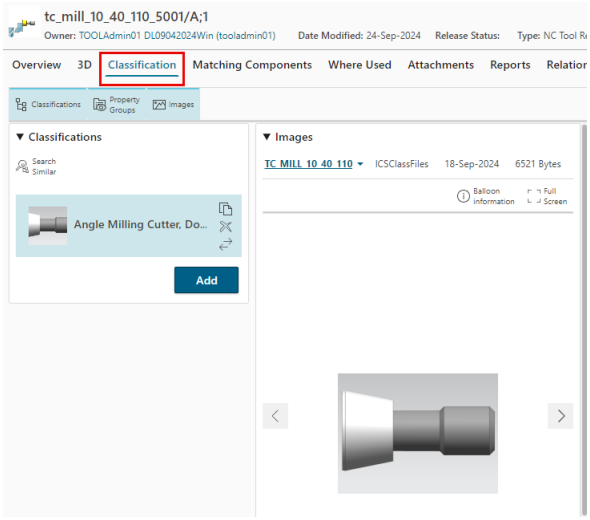
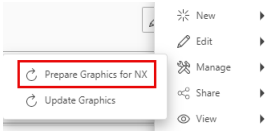
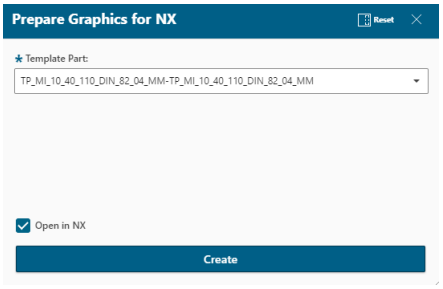


4. Select **More commands** •••→**Manage** →**Prepare Graphics for NX** on the primary toolbar.



5. In the **Prepare Graphics for NX** dialog box, select the template in **Template Part:**, and optionally, select the **Open in NX** check box to open and complete importing the graphic in NX.



> Note:
>
> After preparing the graphic in Active Workspace, you must then use NX to complete the graphic creation process. For more information, see Managing tool assemblies between Manufacturing Resource Library and NX.

6. Click **Create**.

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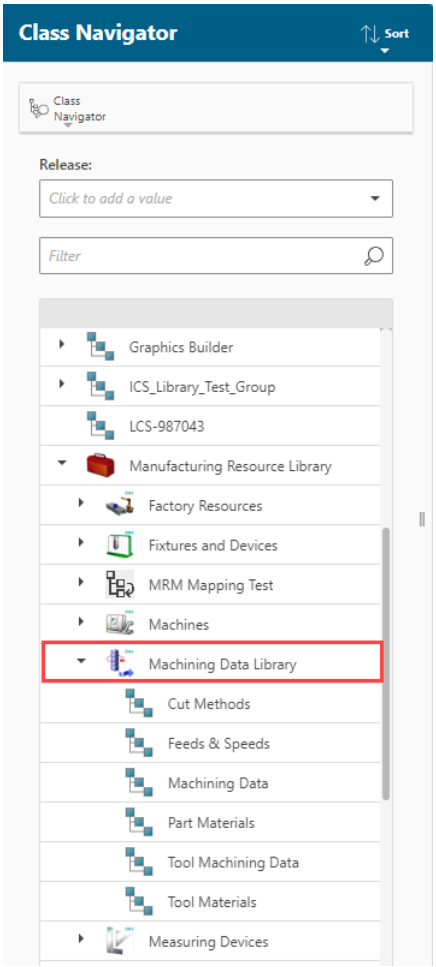
Getting started in the Machining Data Library

Using the Machining Data Library

Use the Machining Data Library to view tool, machine, and material data, and to manage data and reference tables for specific tools, machines, and materials.

Machining Data Library is a user interface in **Classification→Class Navigator→Manufacturing Resource Library→Machining Data Library** that you can use to manage data and reference tables for specific tools, machines, and materials. You can manage cut methods, the material of the cutting tools, and the type of material to be cut.



The **Machining Data Library** includes machining data, and feeds and speeds values, along with classifications for materials, cut methods, and machining data.

You can store machining information in the Machining Data Library using a combination of tool, machine, material and cut method data.

Data tables allow you to view feeds and speeds information.

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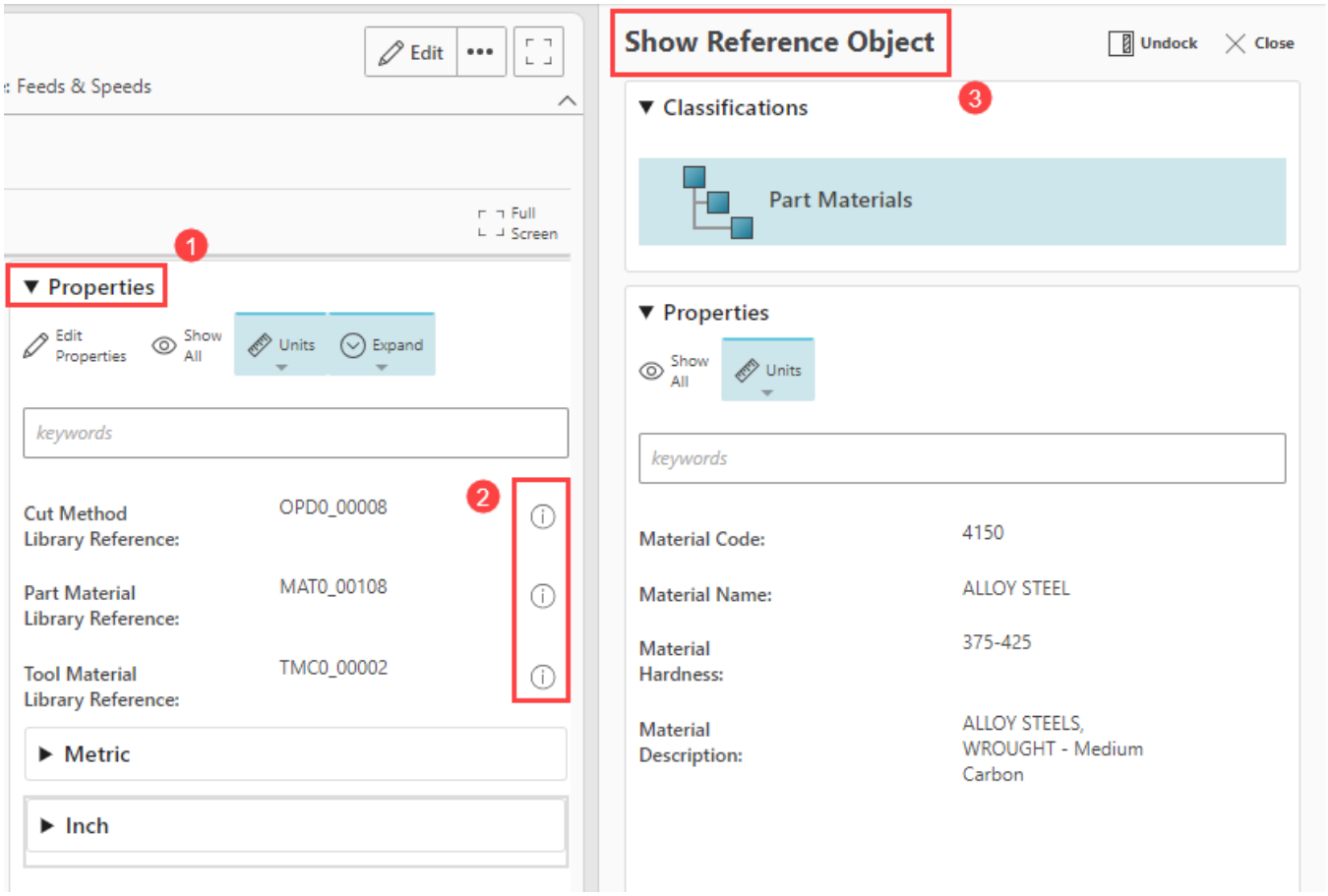
In the (1) **Properties** panel, use (2) **Show Reference Object** ⓘ to view additional information in the (3) **Show Reference Object** panel.

> Note:
>
> An attribute with **Show Reference Object** ⓘ indicates the information is from another reference table and can be edited. For more information, see the topics in Edit data table attributes in Machining Data Library.



> Tip:
>
> Use **Show All** ⓞ in the **Properties** panel to view all available data and reference attributes.

Data tables that contain feeds and speeds information include the following:

- **Feeds & Speeds**
- **Machining Data**
- **Tool Machining Data**

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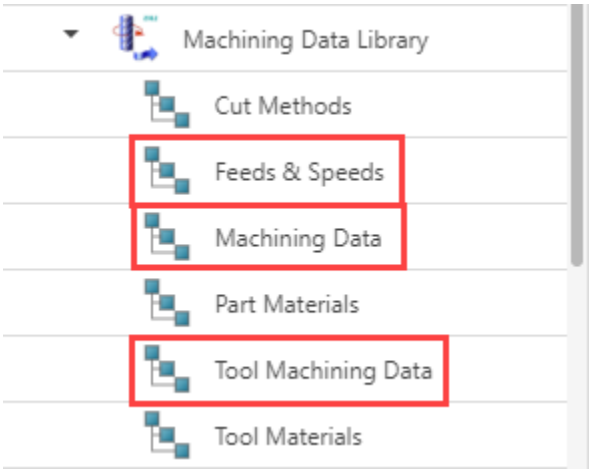

Depending on the data table selected, the following attribute types are available:

Machine Library Reference	All data tables
Cut Method Library Reference	All data tables
Part Material Library Reference	All data tables
Tool Material Library Reference	**Feeds & Speeds** and **Machining Data** tables
Tool Library Reference	**Tool Machining Data** table only

Reference tables contain cut methods and material information that are dependent on other reference tables, and include the following:

- **Cut Methods**
- **Part Materials**
- **Tool Materials**

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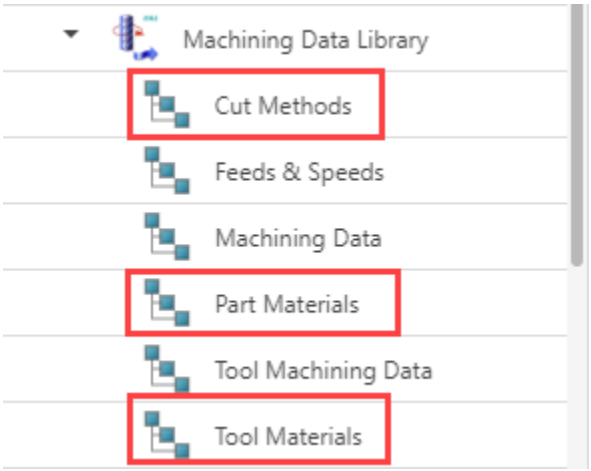

Depending on the reference table selected, the following attribute types are available:

Material Code	**Part Materials** and **Tool Machining Data** tables
Material Name	**Part Materials** and **Tool Machining Data** tables
Material Hardness	**Part Materials** and **Tool Machining Data** tables
Machine Mode	**Cut Methods** table only
Name of Cutting Method	**Cut Methods** table only
Material Description	All reference tables

> Note:
>
> Attributes in reference tables can be edited. For more information, see **Edit reference table attributes in Machining Data Library**.

Edit data table attributes in Machining Data Library

Edit the Machine Library Reference data table attribute

In the **Machine Library Reference** data table, you can assign or reassign a classification attribute in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

Procedure

1. Navigate to the **Machine Library Reference** data table in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

   Classification→Class Navigator→Manufacturing Resource Library→Machining Data Library.

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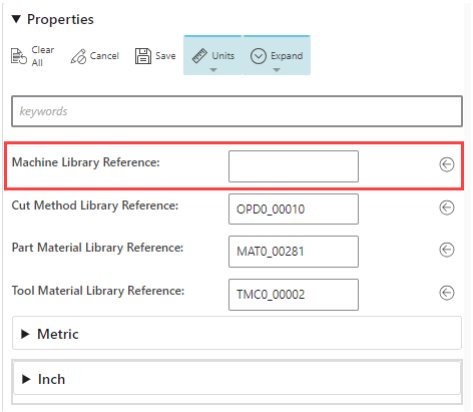
> Note:
>
> Data tables contain tool, machine, and material information located in **Feeds & Speeds**, **Machining Data**, and **Tool Machining Data**.

2. In the **Properties** panel, click **Edit Properties**.

3. Click **Assign Reference Object** next to the **Machine Library Reference** attribute.



The **Assign** panel is displayed.

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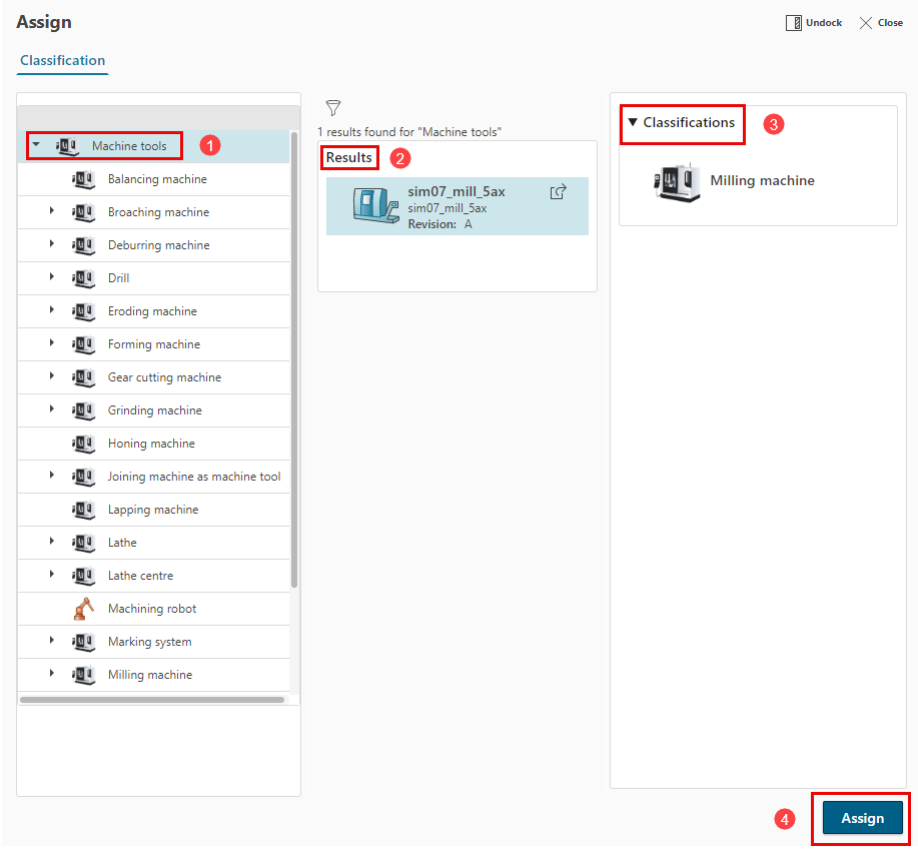

Note:

Only the (1) **Machine tools** classification types are displayed.

4. Select the item classification type on the **Assign** panel.

    a. (Optional) Open the classification types and select the desired item.

    b. Select the item on the (2) **Results** panel.

    c. (Optional) Use the (3) **Classifications** panel to verify the classification of the item.

    d. Click (4) **Assign**.

5. Save the attribute assignment in the **Properties** panel.

Edit the Cut Method Library Reference data table attribute

In the **Cut Method Library Reference** data table, you can assign or reassign a classification attribute in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

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Procedure

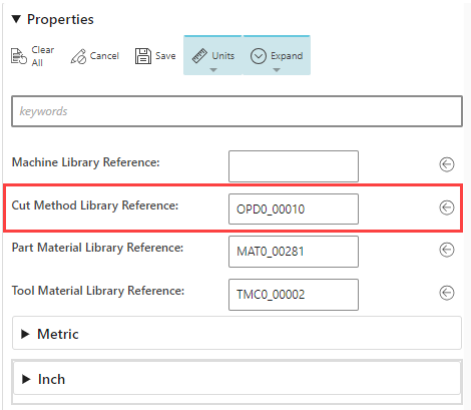
1. Navigate to the **Cut Method Library Reference** data table in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

   Classification→Class Navigator→Manufacturing Resource Library→Machining Data Library.

   > Note:
   >
   > Data tables contain tool, machine, and material information located in **Feeds & Speeds**, **Machining Data**, and **Tool Machining Data**.

2. In the **Properties** panel, click **Edit Properties**.

3. Click **Assign Reference Object** next to the **Cut Method Library Reference** attribute.



   The **Assign** panel is displayed.

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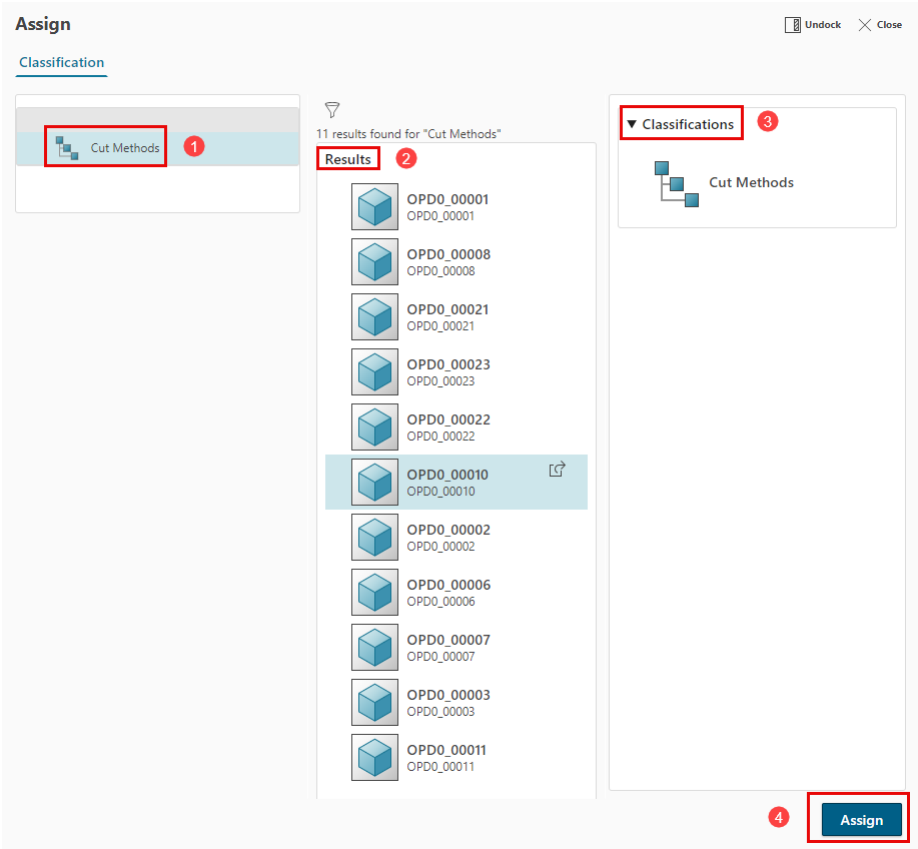

Note:

Only the (1) **Cut Methods** classification types are displayed.

4. Select the item classification type on the **Assign** panel.

   a. Select the item on the (2) **Results** panel.

   b. (Optional) Use the (3) **Classifications** panel to verify the classification of the item.

   c. Click (4) **Assign**.

5. Save the attribute assignment in the **Properties** panel.

Edit the Part Material Library Reference data table attribute

In the **Part Material Library Reference** data table, you can assign or reassign a classification attribute in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

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Procedure

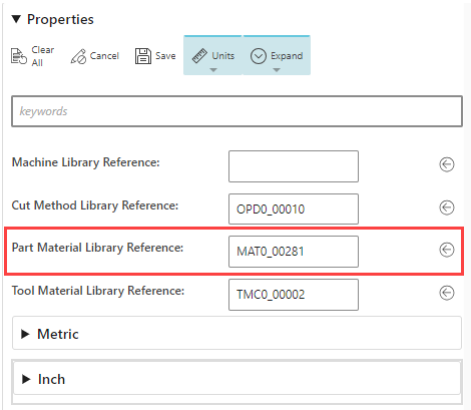
1. Navigate to the **Part Material Library Reference** data table in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

   Classification→Class Navigator→Manufacturing Resource Library→Machining Data Library.

   > Note:
   >
   > Data tables contain tool, machine, and material information located in **Feeds & Speeds**, **Machining Data**, and **Tool Machining Data**.

2. In the **Properties** panel, click **Edit Properties**.

3. Click **Assign Reference Object** next to the **Part Material Library Reference** attribute.



   The **Assign** panel is displayed.

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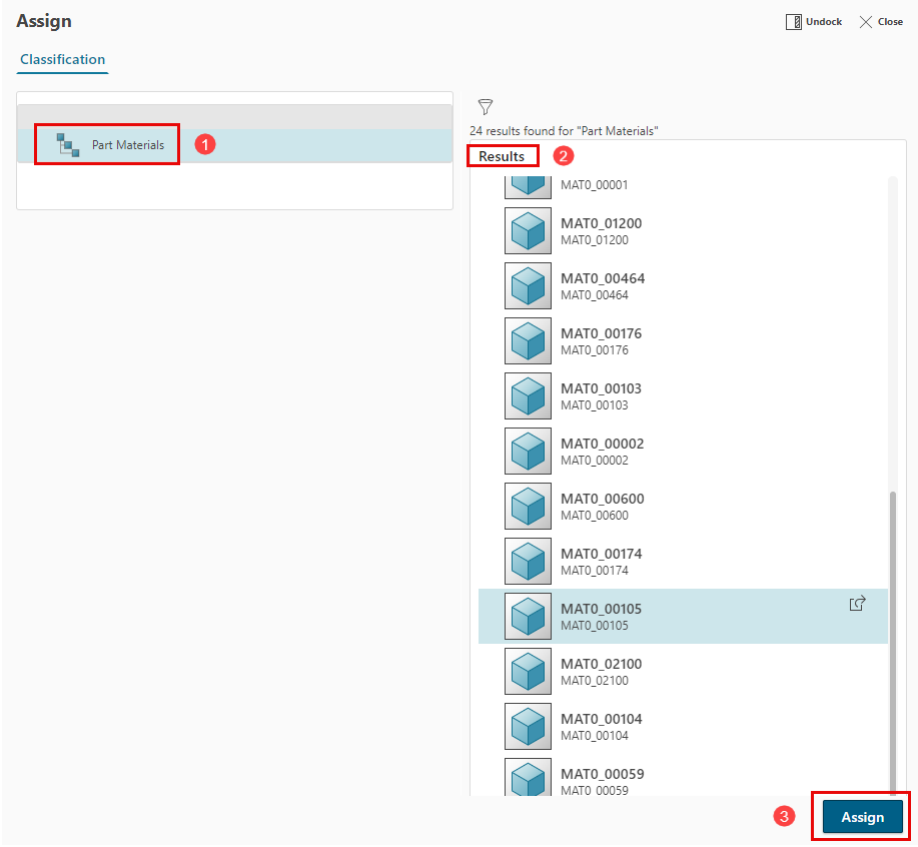

Note:

Only the (1) **Part Materials** classification types are displayed.

4. Select the item classification type on the **Assign** panel.

   a. Select the item on the (2) **Results** panel.

   b. Click (3) **Assign**.

5. Save the attribute assignment in the **Properties** panel.

Edit the Tool Material Library Reference data table attribute

In the **Tool Material Library Reference** data table, you can assign or reassign a classification attribute in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

Procedure

1. Navigate to the **Tool Material Library Reference** data table in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

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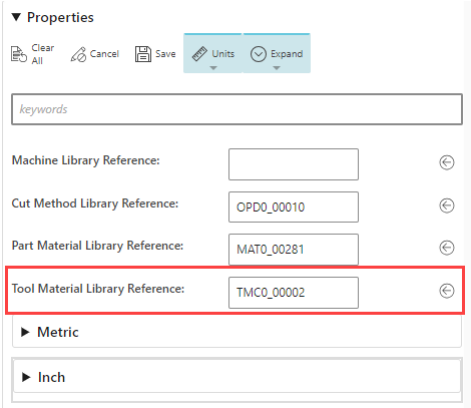
Classification→Class Navigator→Manufacturing Resource Library→Machining Data Library.

> Note:
>
> Data tables contain tool, machine, and material information located in **Feeds & Speeds**, **Machining Data**, and **Tool Machining Data**.

2. In the **Properties** panel, click **Edit Properties**.

3. Click **Assign Reference Object** next to the **Tool Material Library Reference** attribute.



The **Assign** panel is displayed.

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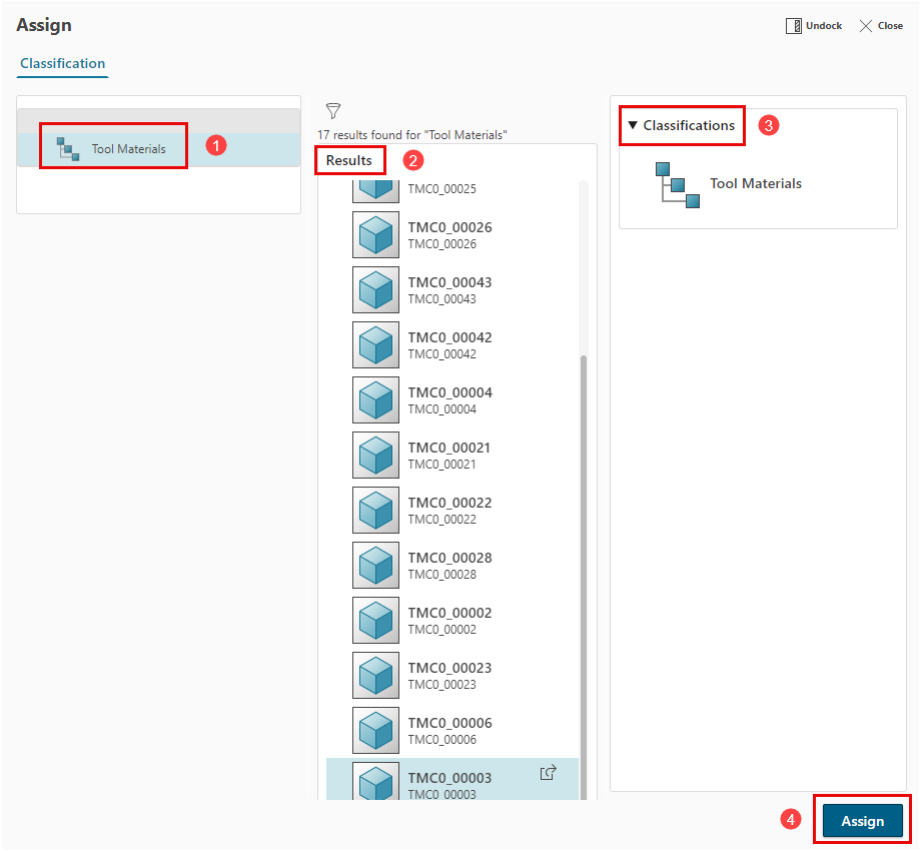

Note:

Only the (1) **Tool Materials** classification types are displayed.

4. Select the item classification type on the **Assign** panel.

   a. Select the item on the (2) **Results** panel.

   b. (Optional) Use the (3) **Classifications** panel to verify the classification of the item.

   c. Click (4) **Assign**.

5. Save the attribute assignment in the **Properties** panel.

Edit the Tool Library Reference data table attribute

In the **Tool Library Reference** data table, you can assign or reassign a classification attribute in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

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Procedure

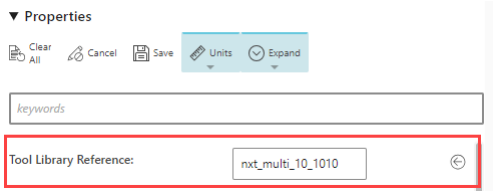
1. Navigate to the **Tool Library Reference** data table in **Feeds & Speeds**, **Machining Data**, or **Tool Machining Data**.

   Classification→Class Navigator→Manufacturing Resource Library→Machining Data Library.

   > Note:
   >
   > Data tables contain tool, machine, and material information located in **Feeds & Speeds**, **Machining Data**, and **Tool Machining Data**.

2. In the **Properties** panel, click **Edit Properties**.

3. Click **Assign Reference Object** next to the **Tool Library Reference** attribute.



   The **Assign** panel is displayed.

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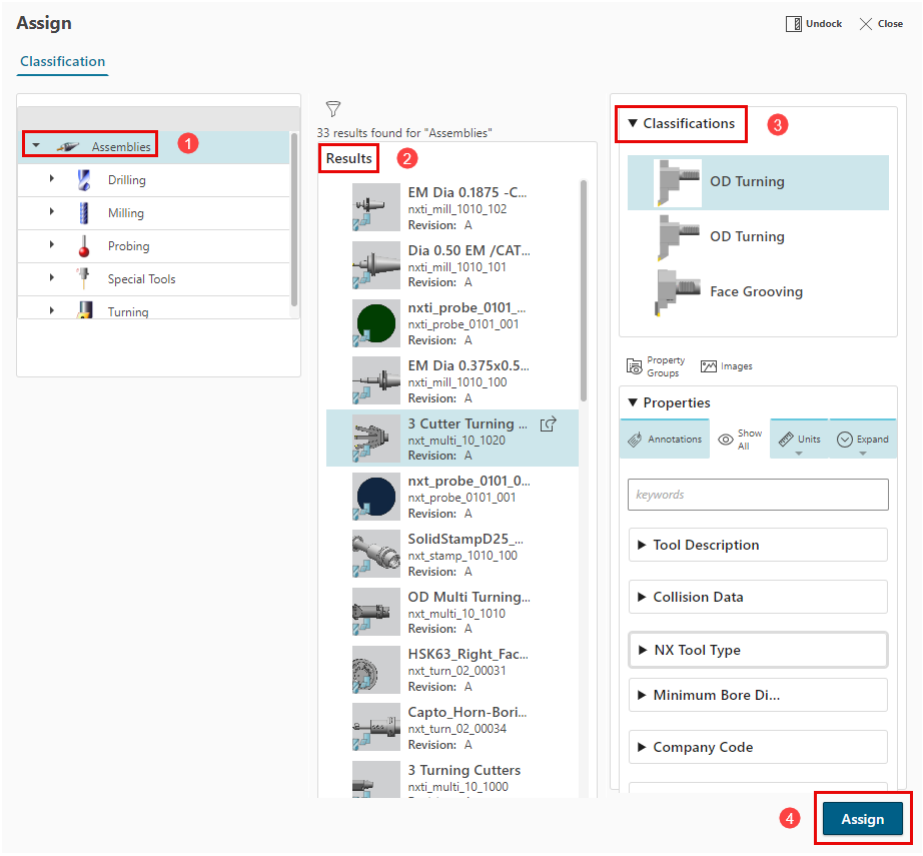

Note:

Only the (1) **Assemblies** classification types are displayed.

4. Select the item classification type on the **Assign** panel.

   a. (Optional) Open the classification types and select the desired item.

   b. Select the item on the (2) **Results** panel.

   c. (Optional) Use the (3) **Classifications** panel to verify the classification of the item.

   d. Click (4) **Assign**.

5. Save the attribute assignment in the **Properties** panel.

Edit reference table attributes in Machining Data Library

In the reference tables, you can add or edit attribute information in **Cut Methods**, **Part Materials**, and **Tool Materials**.

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Procedure

1. Navigate to the desired reference table in **Cut Methods**, **Part Materials**, or **Tool Materials**.

   Classification→Class Navigator→Manufacturing Resource Library→Machining Data Library.

   > Note:
   >
   > Reference tables contain machine, cutting, and material information located in **Cut Methods**, **Part Materials**, and **Tool Materials** classifications.

2. In the **Properties** panel, click **Edit Properties** .

3. Enter the attribute properties and save.

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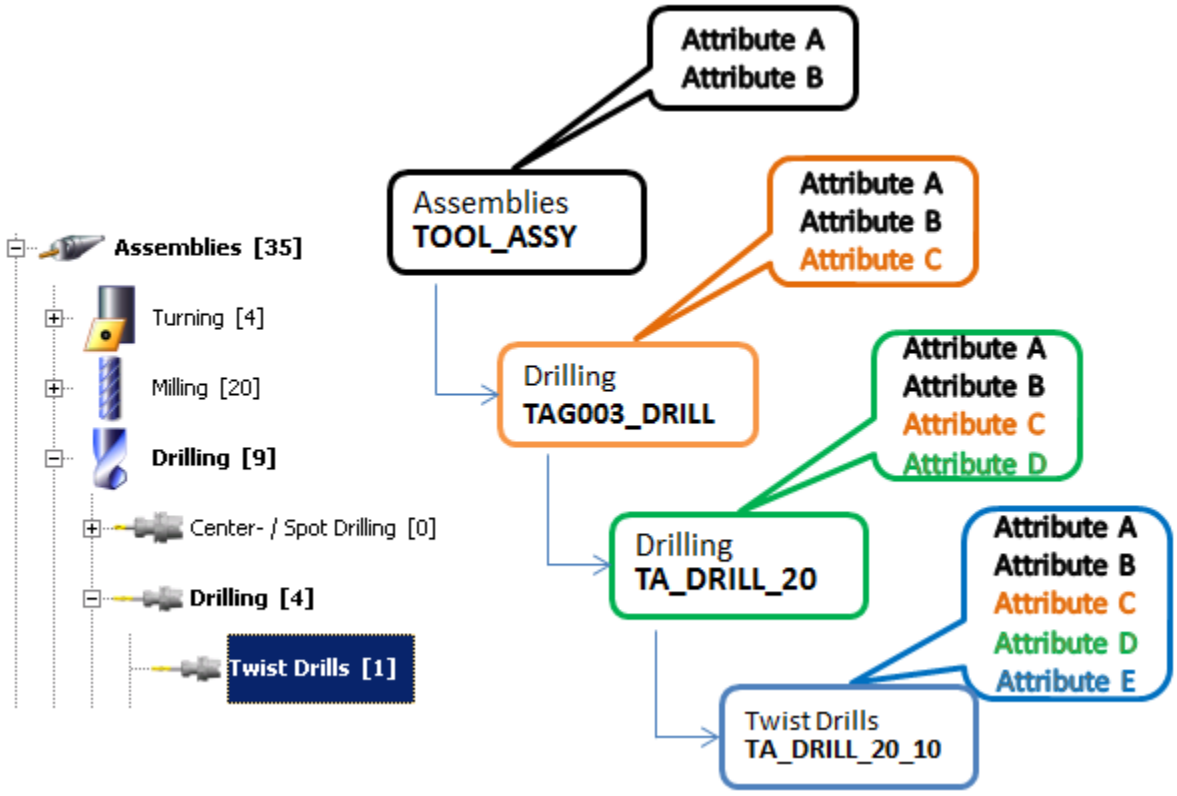
6. Managing tooling information

Applying and inheriting attributes

About attribute inheritance

When creating a classification structure, administrators assign attributes to the different classes in the hierarchy. These attributes describe each class and all subclasses of that class. Each subclass inherits all the attributes that are contained in the parent class. Setting up the attributes in this way makes searching more effective. You can select a top class and search for a tool within that class using a particular value knowing that that attribute is also contained in all the child classes of the class. In the figure below, you can see that only **Attribute E** is assigned to the **Twist Drills** class. All other attributes are inherited from the parent classes.



Edit component or assembly attributes

1. Select the root node or component node that you intend to modify.
2. Select the **Classification** tab.
3. (Optional) To edit the class of the selected resource, click **Edit Class** ✎.

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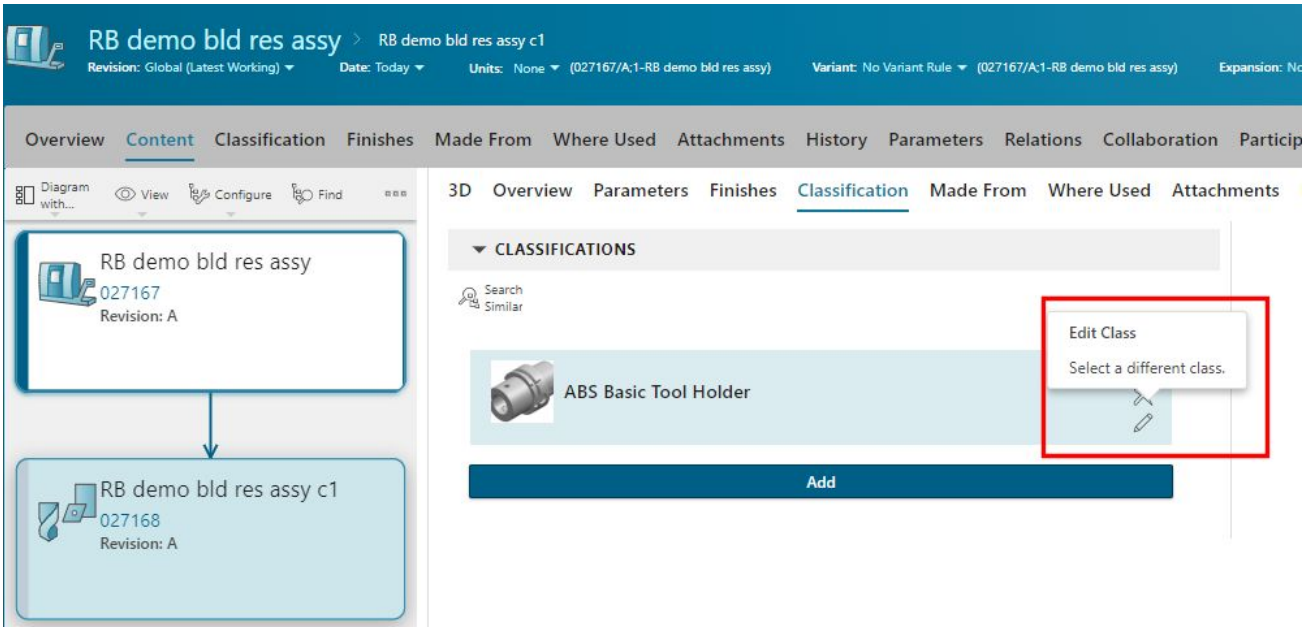
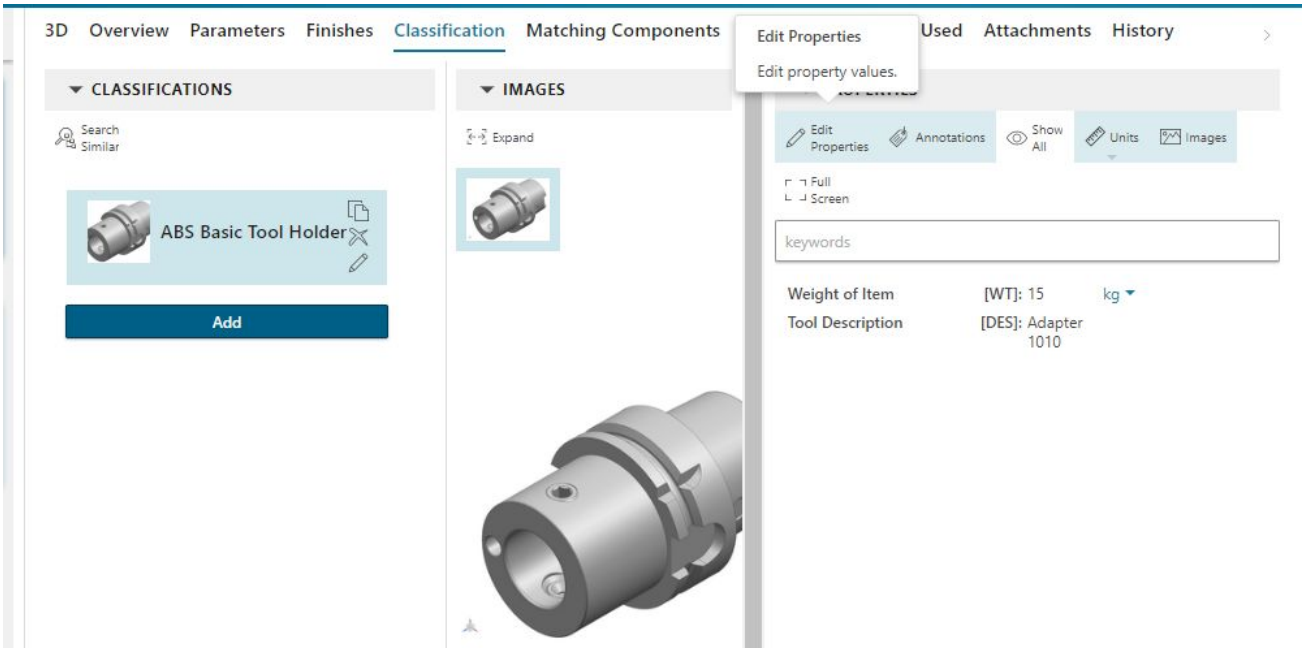
Note:

The available tabs and their order can differ depending on what templates were imported and where you are in Active Workspace.



4. (Optional) To edit the properties of the selected resource, click the **Edit Properties** icon.



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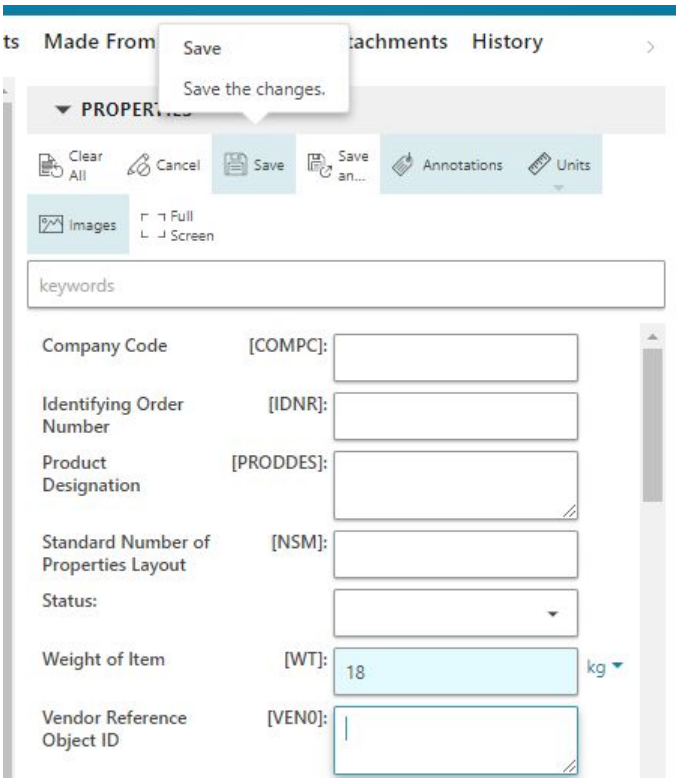
5. Make any necessary changes to the class or properties.

6. Click **Save**.



Add resource classification and data files to another resource

You can copy the classification and data files of a selected resource and add them to another resource.

1. On Active Workspace, click the Manufacturing Resource Library (MRL) dashboard workspace.

2. On the MRL dashboard workspace, select a resource that includes the classification and data files to be added to another resource.

3. Select **More commands** •••→**Manage** →**Map Classification Object** on the primary toolbar.

4. In the **Map Resource** dialog box, in the **TARGET CLASSIFICATION** group, select the target class from the **Select a target class** list, or search for it in the **Search** field.

5. In the **3D MODEL** group, select the **Copy Dataset from Source Item** check box.

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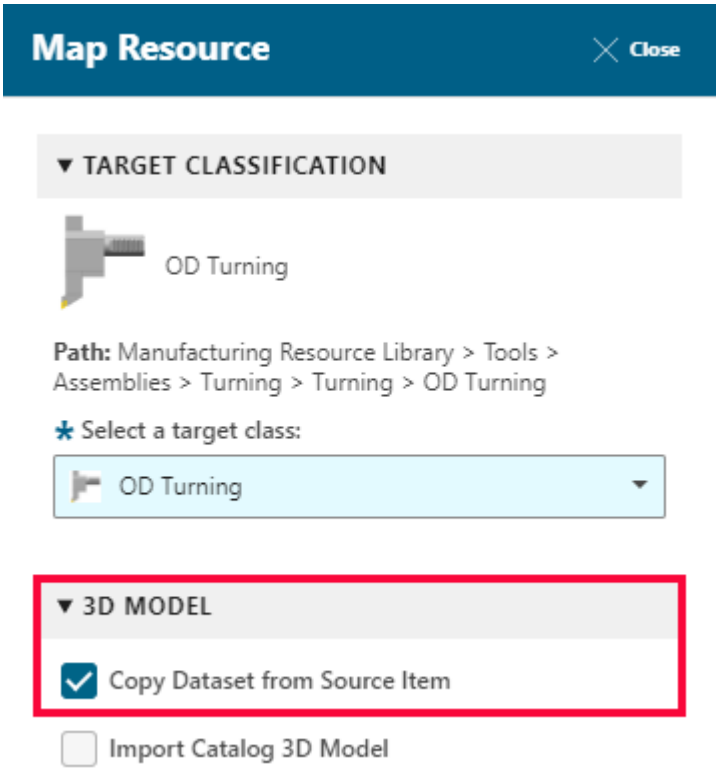

6. Select the resource type from the **TYPE** list, or search for it in the **Search** field.

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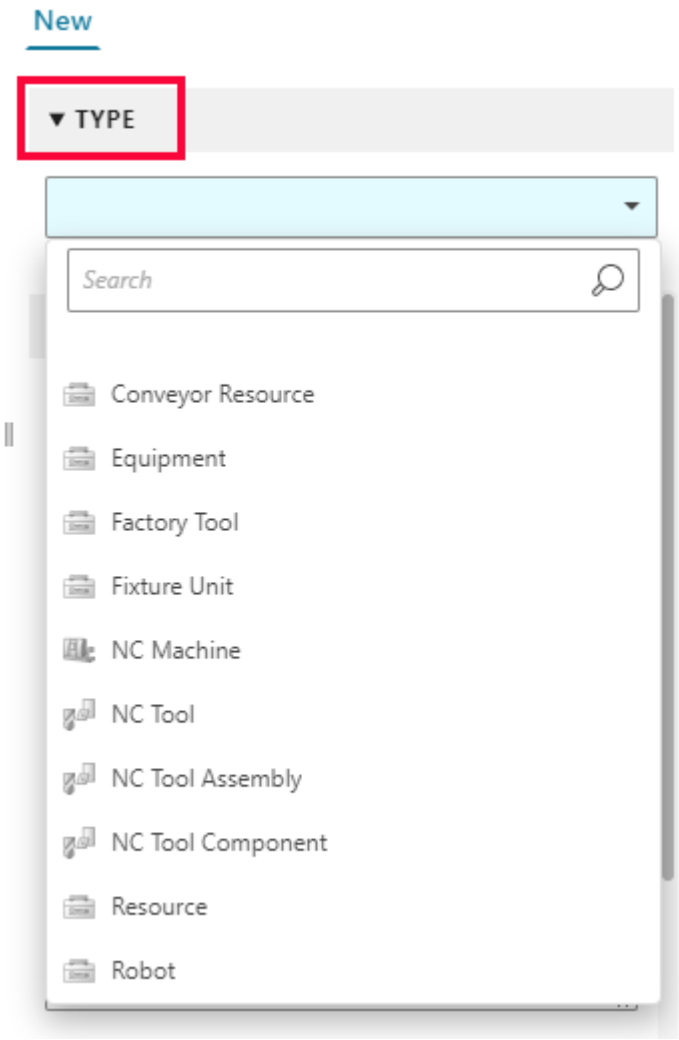

7. Enter the **PROPERTIES** for the resource.

ID	MRL automatically assigns a unique ID that you can accept or change. If you change it, the new ID must be unique.
Revision	If this is a revision of an existing resource, type the next consecutive revision letter.
Name	Type the name of the new resource. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the new resource.

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Propagating component attributes to the resource assembly

About attribute propagation

Attribute propagation allows you to automatically populate the values for an assembly from the structure, instead of manually re-entering component attribute values in each new assembly.

When you assign a propagation start point (PSP), Teamcenter copies attribute values from individual components to the root node of the assembly. You can then focus your Classification searches on the assembly class to find the resource data you need to manufacture a particular process or operation. Because selected component attributes are stored in the assembly class, you do not need to search the entire component list to find the attribute information you need.

A PSP defines where in the resource structure the component attribute propagation begins. You can set a PSP at any level in the resource assembly structure. Teamcenter looks to the assigned PSP component for the attribute value to propagate. If the attribute is not found in that component, the application moves up the structure hierarchy toward the root node one component at a time, until it finds the attribute. After the attribute is found, its value is added to the root node and saved with the assembly.

A resource component assigned as a PSP is identified with the PSP icon . When the component is selected in **Classifications**, the color of the PSP icon changes to yellow to indicate the attribute values for that item are being used.

> Note:
>
> The associated attributes for the **Drill**, identified as the PSP, will propagate to the **Twist Drill** tool assembly.

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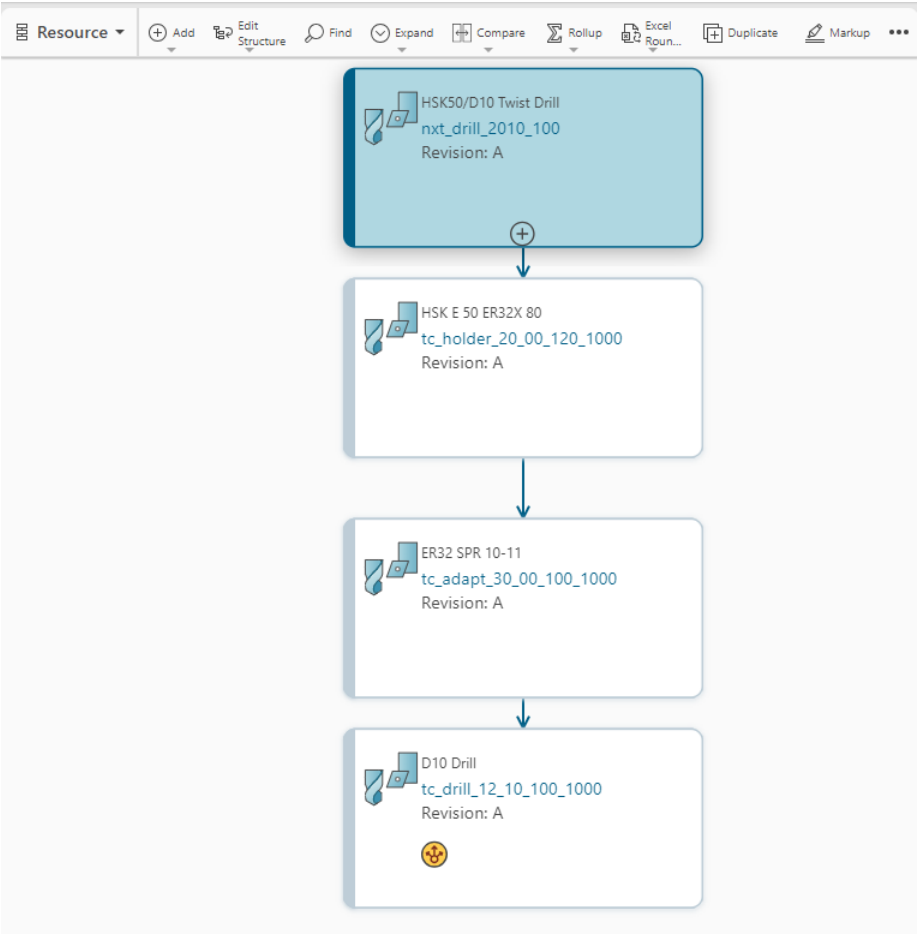

A tool assembly can only have a single PSP assigned to the tool assembly structure. A multitool assembly can have one or more PSPs assigned to the tool assembly structure. A Root node is selected (1).

- Icon indicates that a PSP with a single classification is assigned to the tool assembly (2).

  Used with a tool or multitool assembly.

- Icon indicates that a PSP with multiple classifications is assigned to the multitool assembly, but the classification is not selected in the **Classifications** panel (3).

  Used with a multitool assembly.

- Icon indicates that a PSP with multiple classifications is assigned to the multitool assembly, and the classification is selected in the **Classifications** panel (4).

  Used with a multitool assembly.

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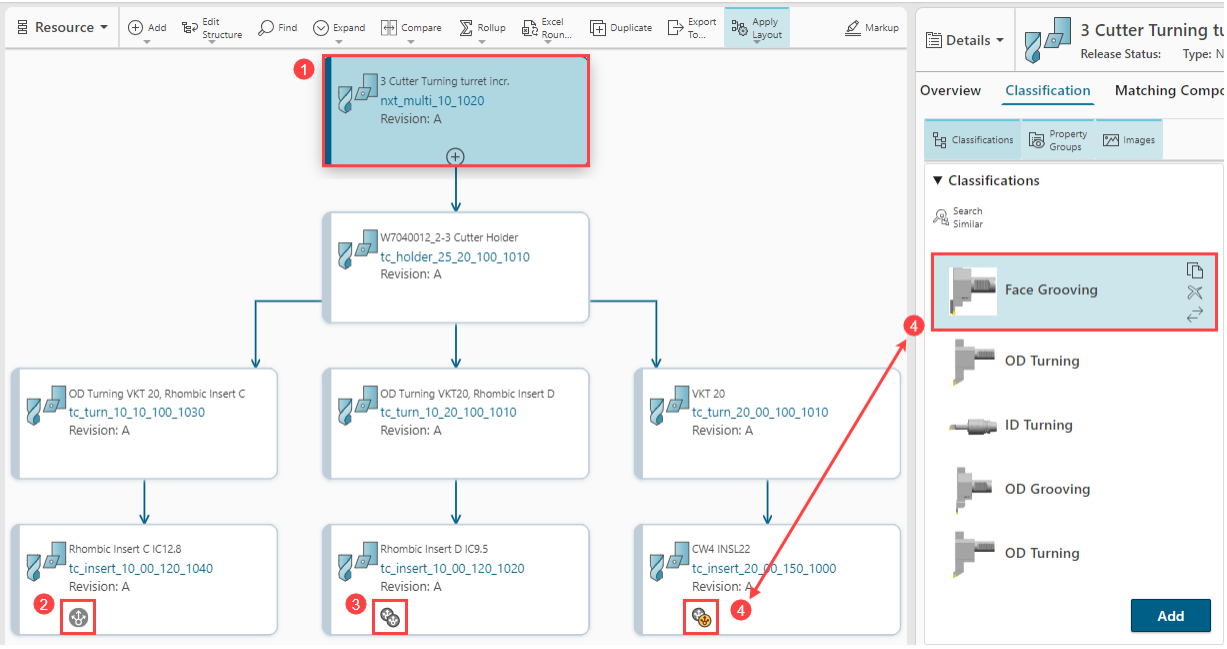

- For more information on assigning a PSP to a tool assembly, see **Add a propagation start point to a tool**.

- For more information on assigning one or more PSPs to a multitool assembly, see **Add an initial propagation start point to a multitool** or **Add additional propagation start points to a multitool**.

Using the PSP attribute values, Teamcenter identifies and populates the attribute values for the tool assembly. Propagated attributes used for the resource assembly structure are identified with an icon next to the attribute in the **Properties** view.

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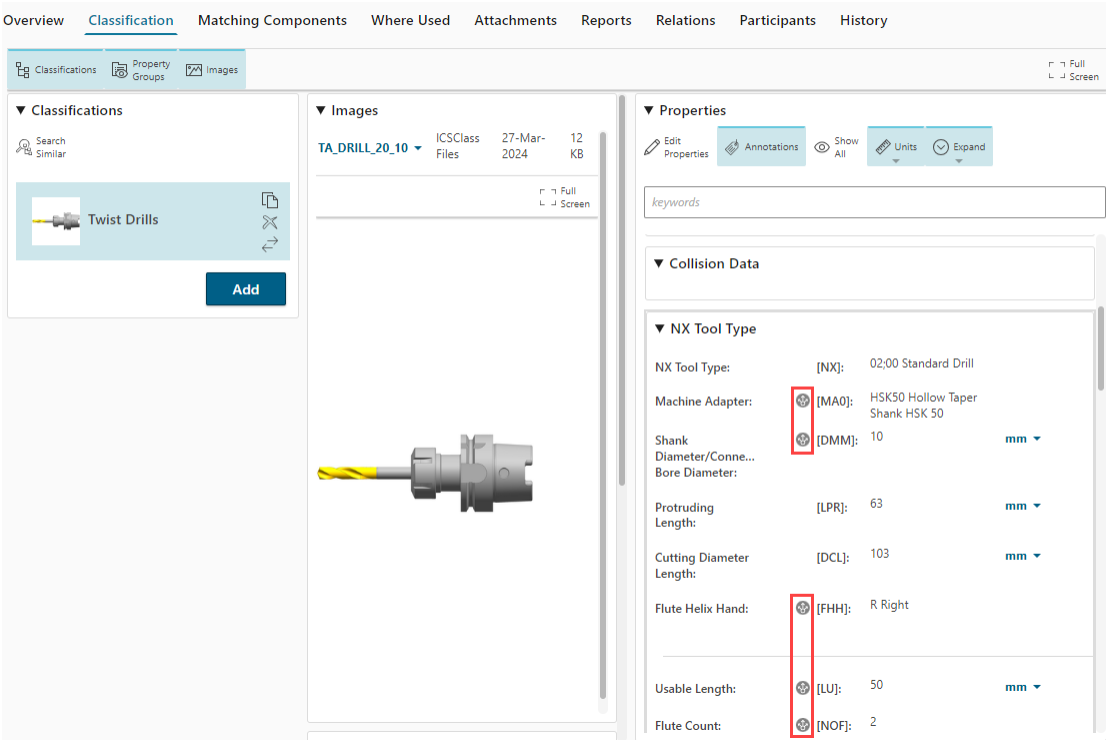

Note:

The **MRMPropagationAlternateAttributes** preference allows you to specify alternate attributes for propagation.

In some cases, the highest component in the structure contains the attribute that should be propagated to other components in the structure. You can use a particular preference to identify specific attributes that should propagate downward rather than upward. For more information see **About reverse attribute propagation**.

Understanding propagated attributes

The following table demonstrates attribute propagation. In this example, the user, while in Classification Admin, specifies the **Test Assembly 1** class as an assembly class. Furthermore, in the assembly class, **Attributes 1–5** have the **Propagated Property** option selected.

Note:

For specific cases that include attributes at the upper levels of a structure that need to be propagated downward, see **About reverse attribute propagation**.

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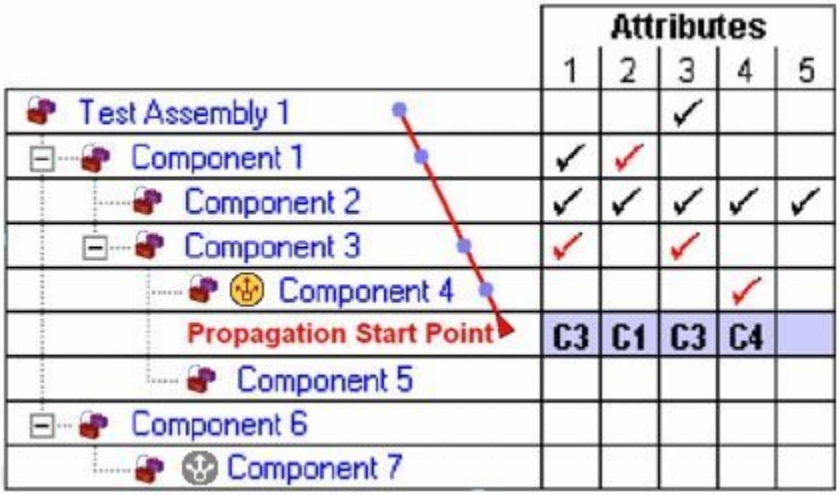

When Teamcenter propagates attributes, it begins at the propagation start point (PSP), and then travels upward through the parent levels looking for attributes to propagate. In the example, the first component it looks at is **Component 4**. The component contains **Attribute 4**, which is a propagated attribute, so Teamcenter propagates **Attribute 4** to the resource assembly, **Test Assembly 1**. Teamcenter then looks at the parent of **Component 4**, which is **Component 3**, and finds two attributes to propagate. It then looks at the parent of **Component 3**, which is **Component 1**, and finds **Attribute 2** to propagate. **Component 1** also contains **Attribute 1**, but this value has already been populated by **Component 3**. When Teamcenter is finished with propagating attributes, the top-level resource assembly, **Test Assembly 1**, shows the following attribute values:

Attribute	Propagated from
Attribute 1	**Component 3**
Attribute 2	**Component 1**
Attribute 3	**Component 3**
Attribute 4	**Component 4**

Attribute 5 contains no value. Although there is a value assigned to this attribute in **Component 2**, **Component 2** does not lie within the propagation path as it is not a parent or grandparent of **Component 4**, the PSP. Similarly, **Components 6** and **7** do not lie within the propagation path.

A gray PSP indicates that this PSP is defined in the context of a subassembly. It is not taken into consideration in the current tool assembly. Subassemblies are highlighted with a colored background in Resource Manager.

Add a propagation start point to a tool

You can add a propagation start point to a tool to automatically populate the values for an assembly.

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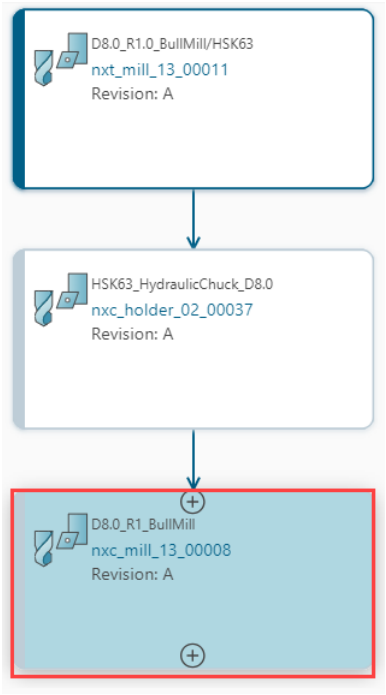
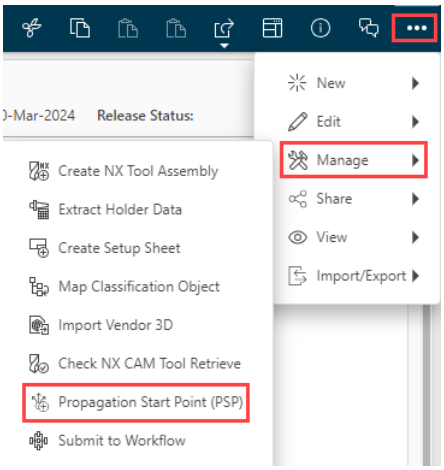
Procedure

1. Select the component in the hierarchy to which you are adding a propagation start point.



2. Select **More commands •••→Manage →Propagation Start Point (PSP)** on the primary toolbar.



A message box confirms the propagation start point is added.

A propagation start point icon is added to the selected component.

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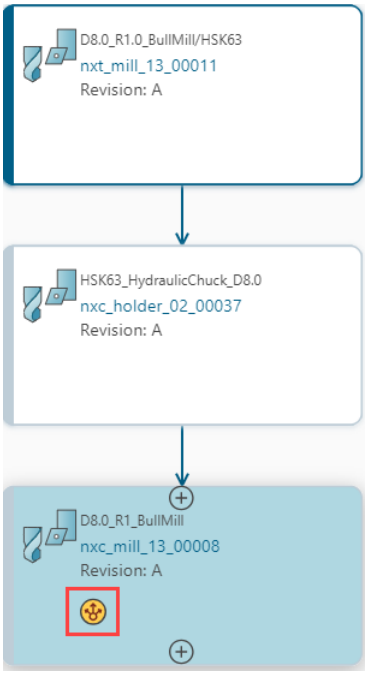

Add an initial propagation start point to a multitool

Add a propagation start point to a multitool to automatically populate the values for an assembly.

Procedure

1. Select the component in the hierarchy to which you are adding a propagation start point.

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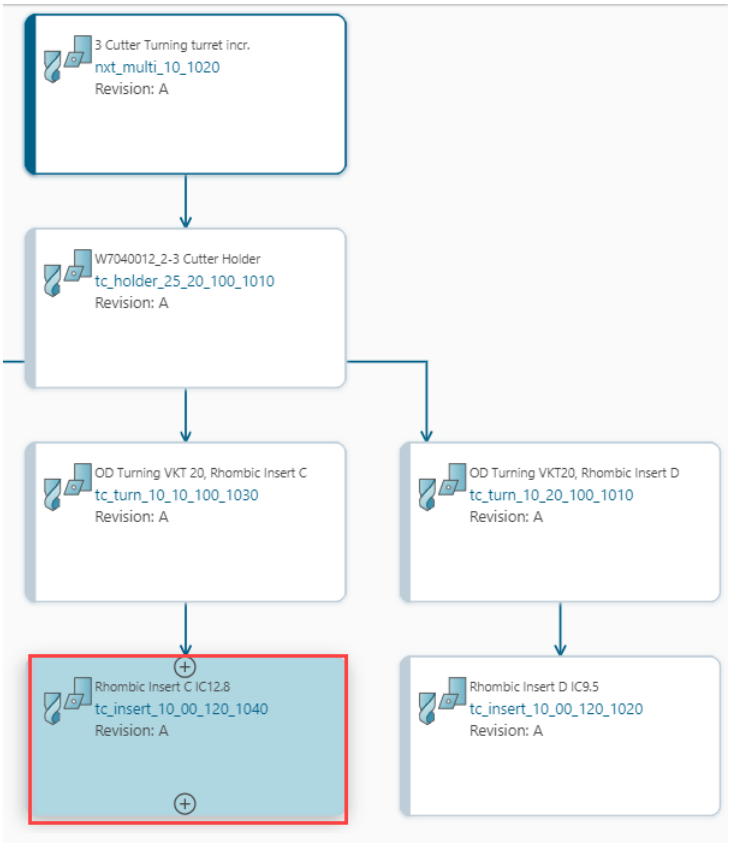
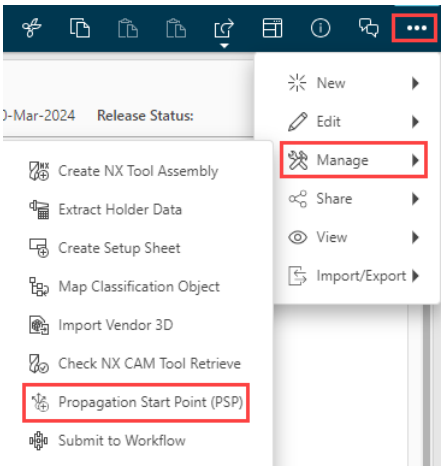


2. Select **More commands •••→Manage →Propagation Start Point (PSP)** on the primary toolbar.



3. Select a classification in the **Propagation Start Point (PSP)** dialog box and click **Add**.

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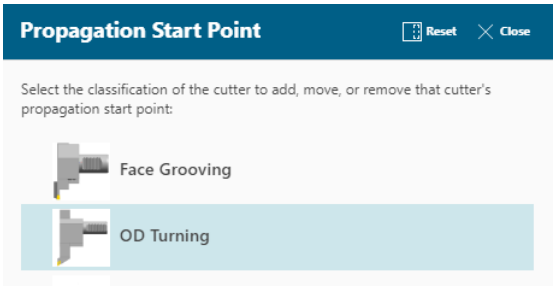
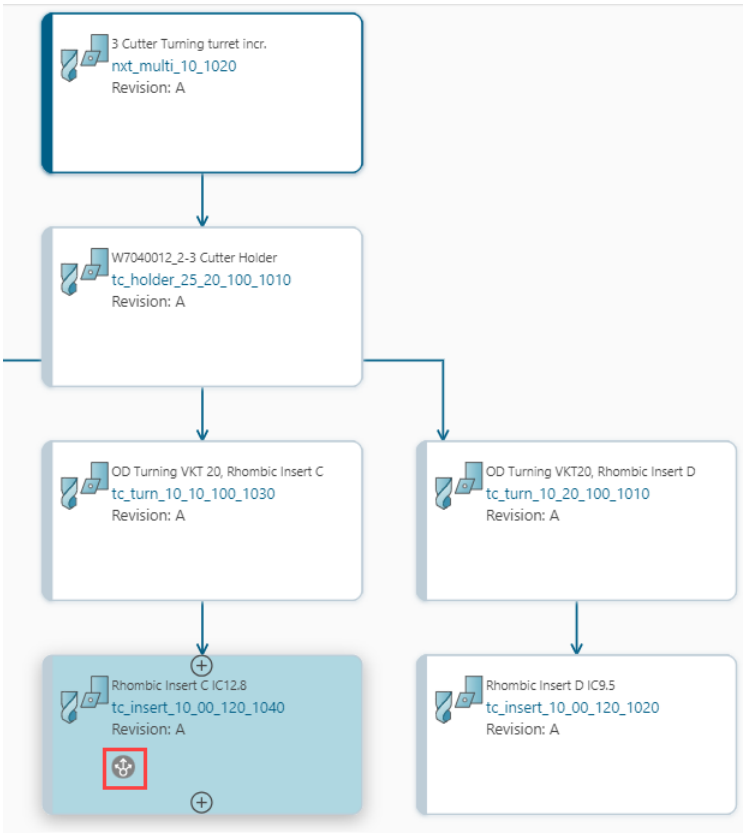


A message box confirms the propagation start point is added.

A propagation start point icon is added to the selected component.



Add additional propagation start points to a multitool

Add multiple propagation start points to a multitool to automatically populate the values for an assembly.

Procedure

1. Select the component in the hierarchy to which you are adding a propagation start point.

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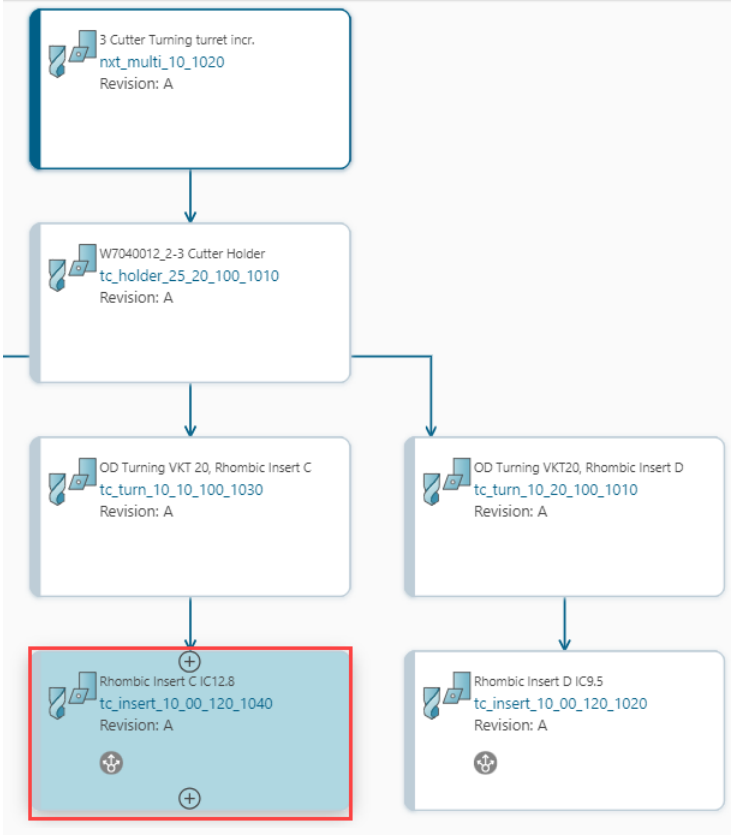
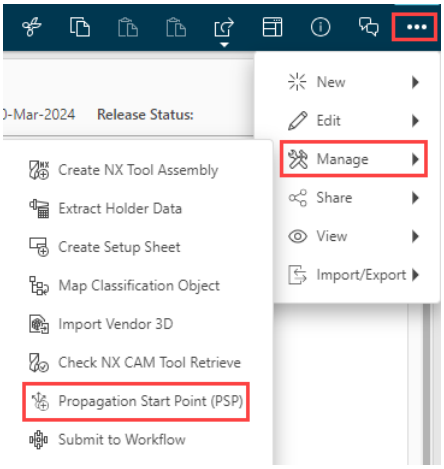


2. Select **More commands** •••→**Manage** →**Propagation Start Point (PSP)** on the primary toolbar.



3. Select a classification in the **Propagation Start Point (PSP)** dialog box and click **Add**.

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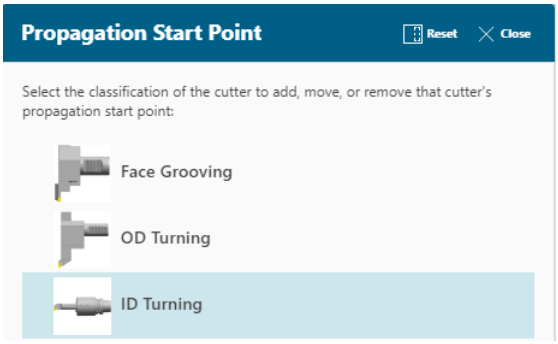
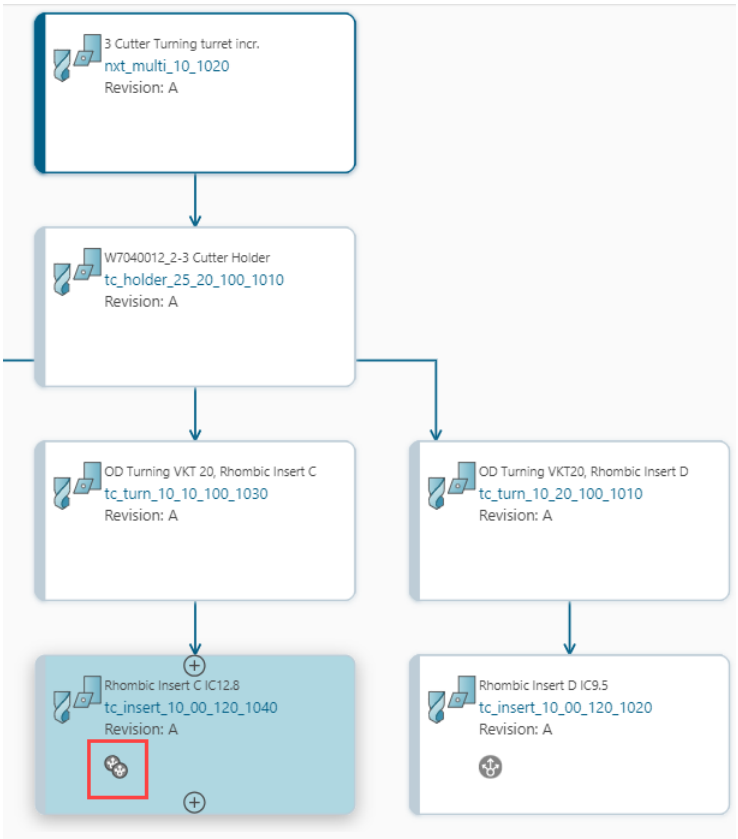


A message box confirms the propagation start point is added.

A multiple propagation start points icon is added to the selected component.



Remove a propagation start point

1. Select the component in the hierarchy from which you are removing a propagation start point.
2. Click **Propagation Start Point** in the **One Click Commands** area.

   Resource Manager prompts you to confirm the removal of the **Propagation Start Point**.

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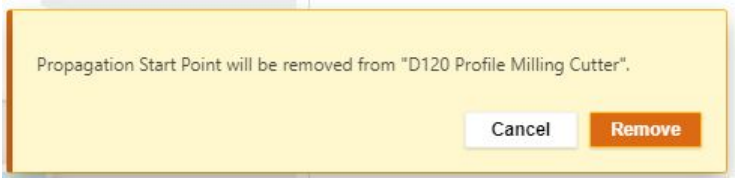


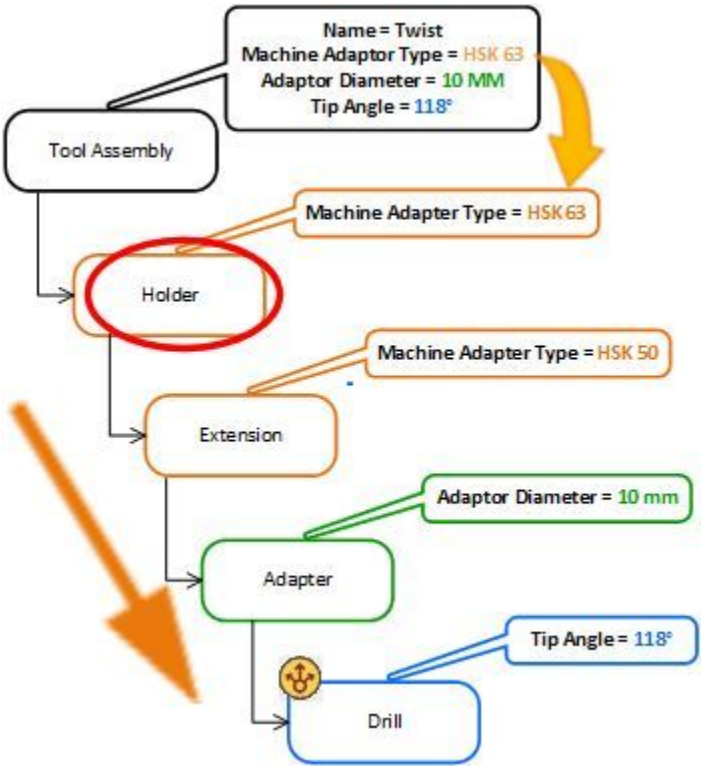
3. Select **Remove**.

   The **Propagation Start Point** indicator is removed from the selected component.

About reverse attribute propagation

By default, assembly attributes are propagated bottom-up. Starting from the propagation start point (PSP) , the components are checked up to the root component to find specific attribute values. The first component closest to the PSP provides the value applied to each component above it in the structure. Reverse propagation works in a similar way, but the attributes are propagated downward from the first component after the PSP.

You can control which attributes are propagated downward by adding the attribute title to the preference **MRMPropagationReverseAttributes.** All attributes listed in the preference propagate in the opposite order: For these attributes, the system checks from the component closest to the root component down the structure to the component with the PSP. In this case, the value of the component closest to the root are propagated downward.



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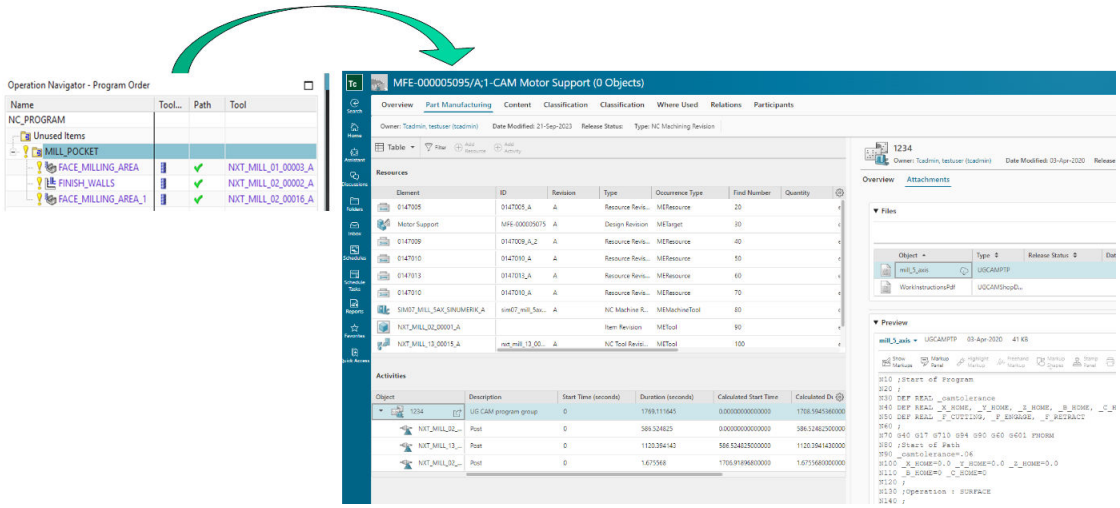
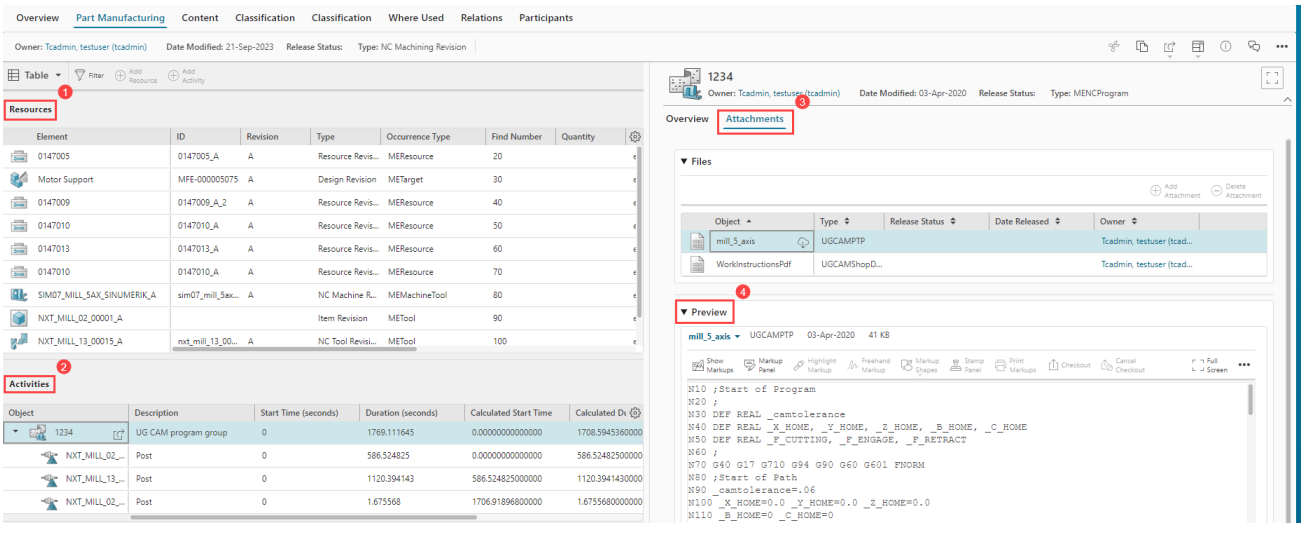
7. Managing manufacturing data with Part Manufacturing

Getting started with Part Manufacturing

Part Manufacturing is a user interface that allows you to create and manage data for NX applications.



Using **Part Manufacturing**, you can access an overview of resources and activity data that is generated by the NC programmer, such as NC programs, shop floor documentation, and NC Machining operations, to manage a CAM setup.



- (1) **Resources**—Displays the machining operation tools, fixtures, and machines that are assigned to the Part Manufacturing Operation, such as NC Machining.

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- (2) **Activities**—Displays the activity groups, individual activities, tasks, and duration of the assigned tools for each machining program. The NC Program file is also displayed.

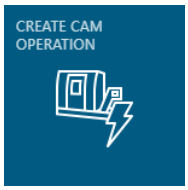
  You can also download activity files from the **Activities** panel.

  > Note:
  >
  > You can edit certain values in the **Activities** panel. Double-click a text or value in an activity row.

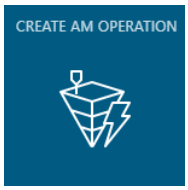
- (3) **Attachments**—Displays the attachments available for the resource or activity that you select.

- (4) **Preview**—Displays a content preview of the file that is selected on the **Attachments** panel, such as tool setup sheets or work instructions.

When the **Manufacturing** profile is selected in the **Workspace** menu, you can use one of the following tiles on the Active Workspace home page to quickly create, initiate, or design manufacturing objects for use in **Part Manufacturing**. Unless noted, you do not need to select the item type, which is preselected in the **TYPE** group list.

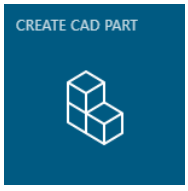


CREATE CAM OPERATION—Use to create an NC Machining operation to manage a CAM setup, or to send to NX CAM to be filled out in detail.



CREATE AM OPERATION—Use to initiate a job task for Additive Manufacturing to plan a job file for 3D printing.

CREATE INSP OPERATION—Use to create an inspection operation that can be used in CMM or NX Metrology.



CREATE CAD PART—Use to design a new CAD part that can be used to manually build an in-stage model.

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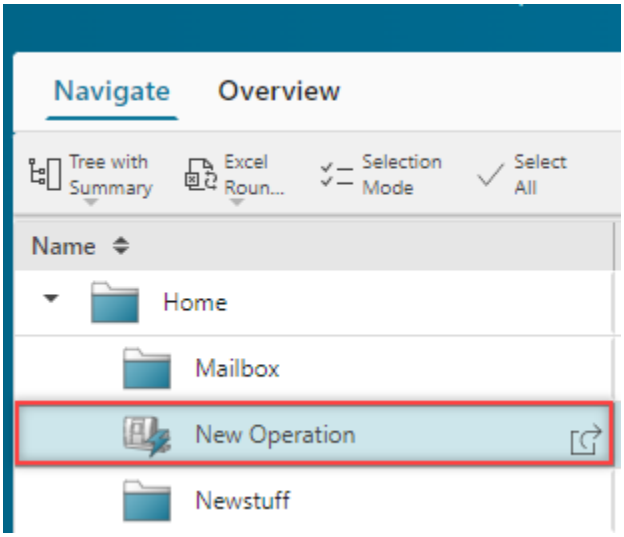
> Note:
>
> You must select the item type in the **TYPE** group list.

CREATE RESOURCE—Use to quickly initiate a new resource item.

After you create an operation, you can locate it in the **Home Folder**. To access the new operation in **Part Manufacturing**, click the **Open** icon and then select the **Part Manufacturing** tab.



Add or remove a resource

Add or remove a resource from the **Resource** panel on the **Part Manufacturing** tab.

Add a resource

1. Select the **Part Manufacturing** tab.

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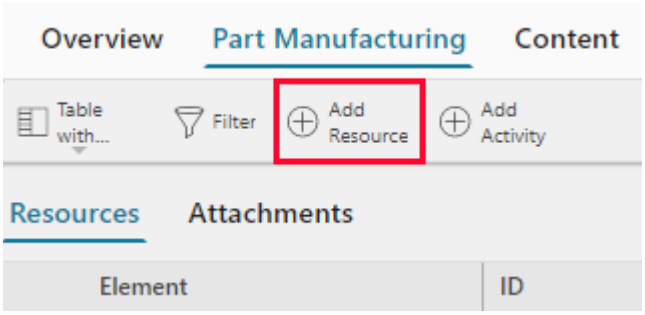
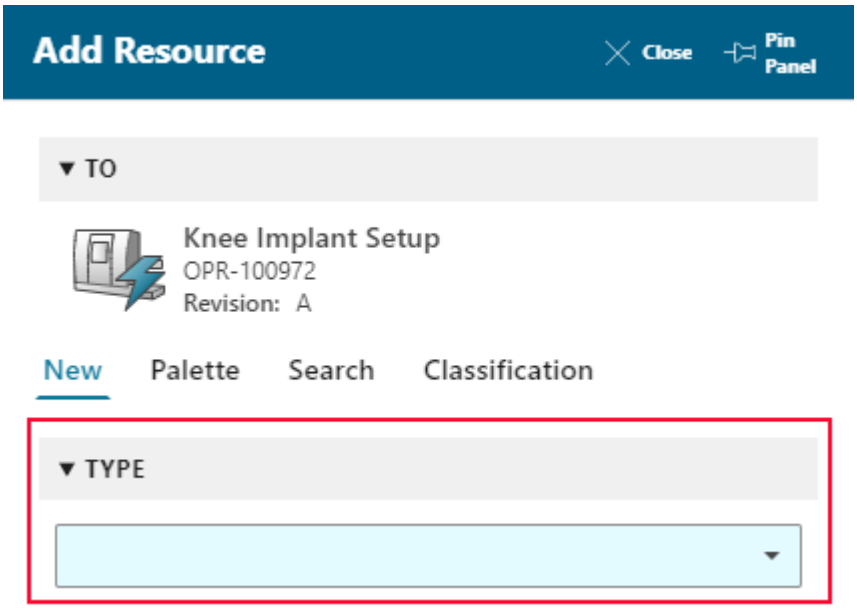


2. Click ⊕ **Add Resource** on the results panel toolbar.

   The **Add Resource** dialog box opens.



3. In the **Add Resource** dialog box, select the resource type from the **TYPE** list and enter the properties.

ID	MRL automatically assigns a unique ID that you can accept or change. If you change it, the new ID must be unique.
Revision	If this is a revision of an existing resource, type the next consecutive revision letter.
Name	Type the name of the new resource. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the new resource.

4. Click **Add**.

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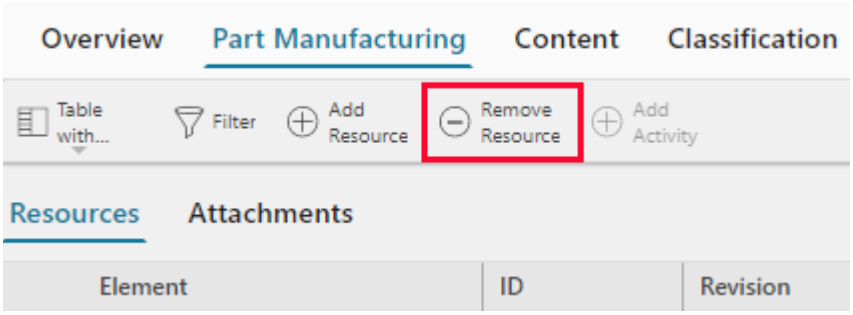
A message box appears confirming the resource is added.

Remove a resource

1. Select the **Part Manufacturing** tab.
2. On the **Resources** panel, in the **Element** column, select a resource.



3. Click ⊖ **Remove Resource** on the results panel toolbar.
4. In the confirmation message box, click **Remove**.

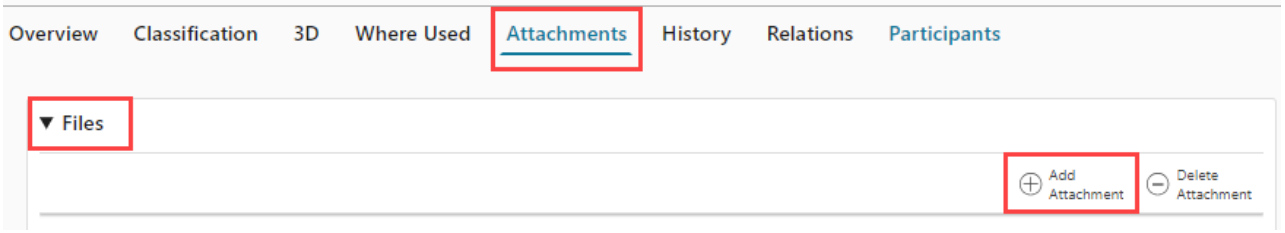
> Note:
>
> The resource is removed from the resource structure only, not from the database.

Add or delete an attachment in an operation or revision

In **Part Manufacturing**, add or delete an attachment in an operation or revision, such as **NC Machining Revision**.

Procedure

1. To add an attachment in an operation or revision, select **Attachments** on the **Part Manufacturing** work area toolbar.



2. Click ⊕ **Add Attachment**.

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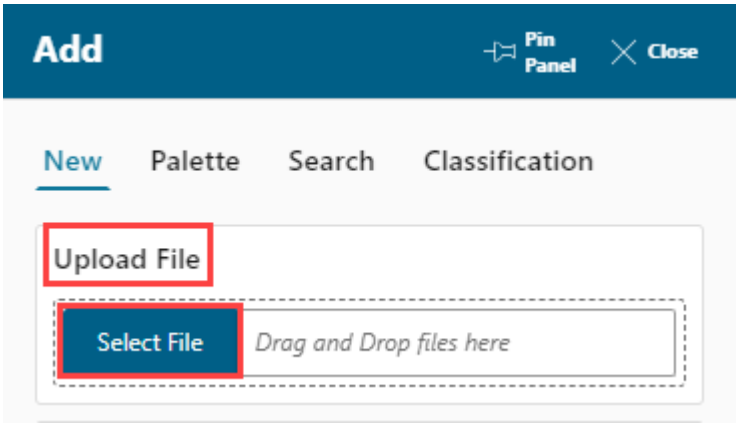
3. In the **Add** dialog box, use **Select File** in **Upload File** to select an item to attach.



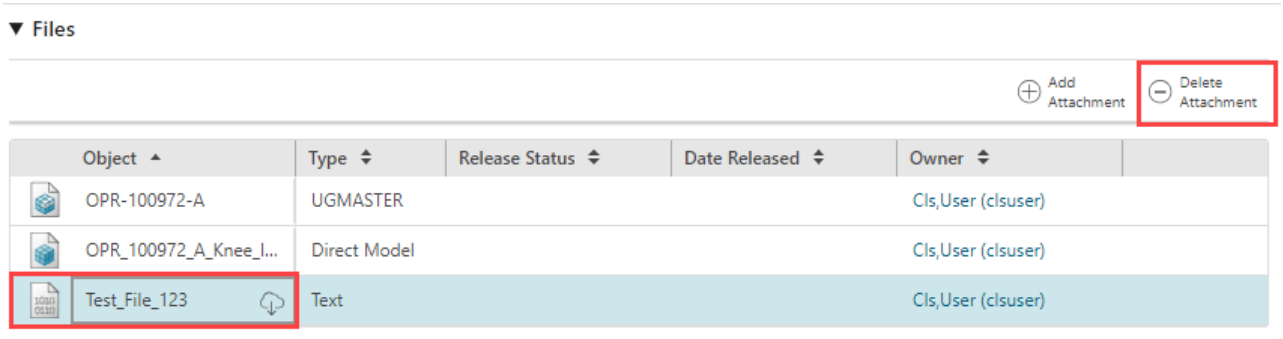
4. Enter the properties for the attachment.

Name	Type the name of the new attachment. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the new resource attachment.
Type	Select the file type for the new attachment.
Relation	(Optional) Select the file relation type for the new attachment.

5. Click **Add**.

   A message confirms the attachment is added.

6. To delete an attachment from an operation or revision, select **Attachments** on the **Part Manufacturing** work area toolbar.



7. On the **Files** panel, select an attachment from the **Object** column.

8. Click ⊖ **Delete Attachment**.

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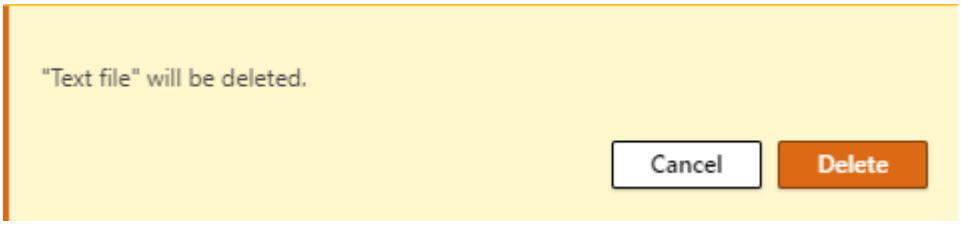
A message appears to confirm you want to delete the attachment.



9. Click **Delete**.

Add or delete an attachment in a resource or activity

In **Part Manufacturing**, add or delete an attachment in a resource or activity item.

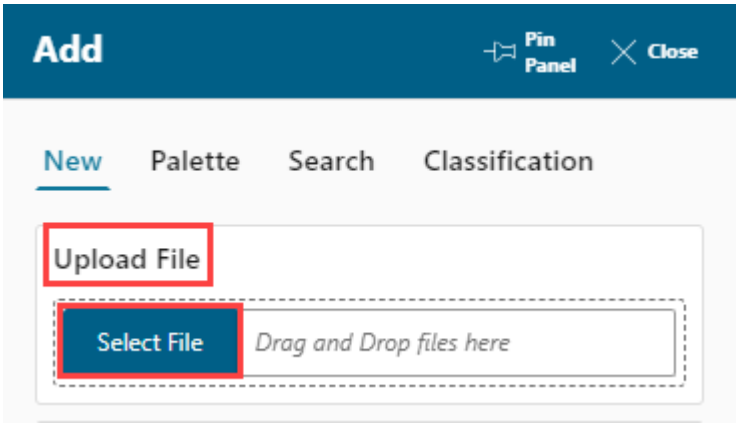
Procedure

1. To add an attachment in a resource or activity, select **Attachments** on the **Part Manufacturing** work area toolbar.

2. Select a resource **Element** or an activity **Object**.

3. Click ⊕ **Add Attachment**.

4. In the **Add** dialog box, use **Select File** in **Upload File** to select an attachment.



5. Enter the properties for the attachment.

Name	Type the name of the new attachment. Use a unique name so it is easier to search the database using this term.
Description	(Optional) Type a description of the new resource attachment.

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Type	Select the file type for the new attachment.
Relation	(Optional) Select the file relation type for the new attachment.

6. Click **Add**.

   A message confirms the attachment is added.

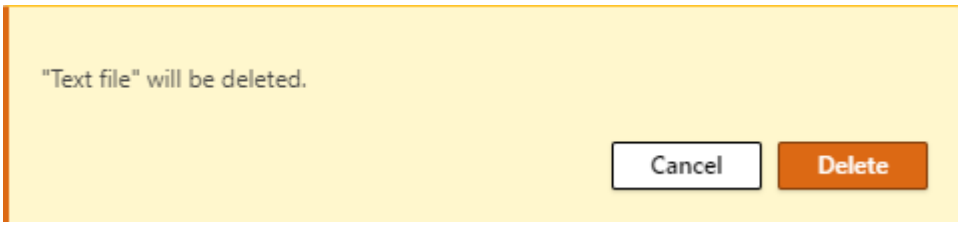
7. To delete an attachment from a resource or activity, select **Attachments** on the **Part Manufacturing** work area toolbar.

8. Select an item on the **Resources** or **activities** panel.

9. On the **Files** panel, select an attachment from the **Object** column.

10. Click ⊖ **Delete Attachment**.

    A message appears to confirm you want to delete the attachment.

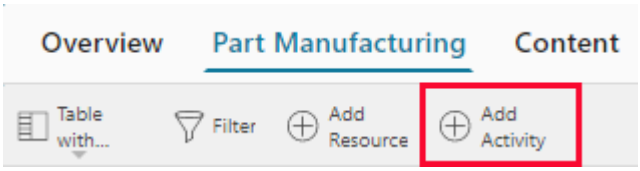


11. Click **Delete**.

Add a parent activity

Add a new parent activity to the **Activities** panel, on the **Part Manufacturing** tab.

1. Select the **Part Manufacturing** tab.



2. Click ⊕ **Add Activity** on the results panel toolbar.

   The **Add Activity** dialog box opens.

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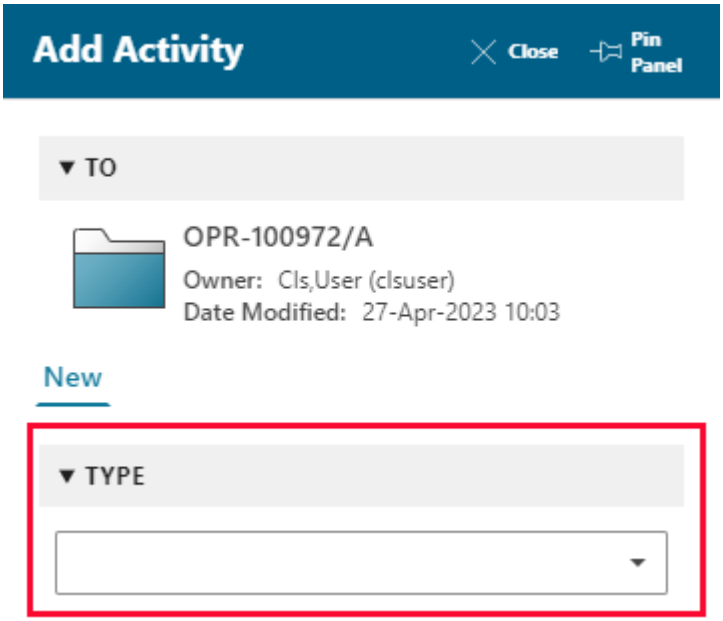


3. In the **Add Activity** dialog box, select the activity type from the **TYPE** list and enter the properties.

Name	Type the name of the new activity. It is recommended that you make the name unique so users can easily search the database using this term.
Description	(Optional) Type a description of the new activity.

4. Click **Add**.

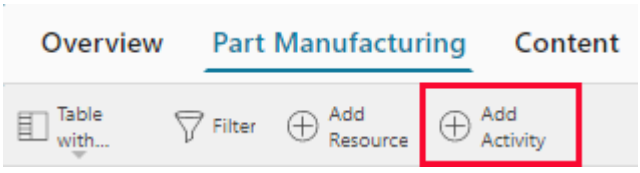
   A message box appears confirming the activity is created.

Add a child activity

Add a new activity to an existing parent or child activity in the **Activities** panel, on the **Part Manufacturing** tab.

1. Select the **Part Manufacturing** tab.

2. Select an activity in the **Object** column on the **Activities** panel.



3. Click ⊕ **Add Activity** on the results panel toolbar.

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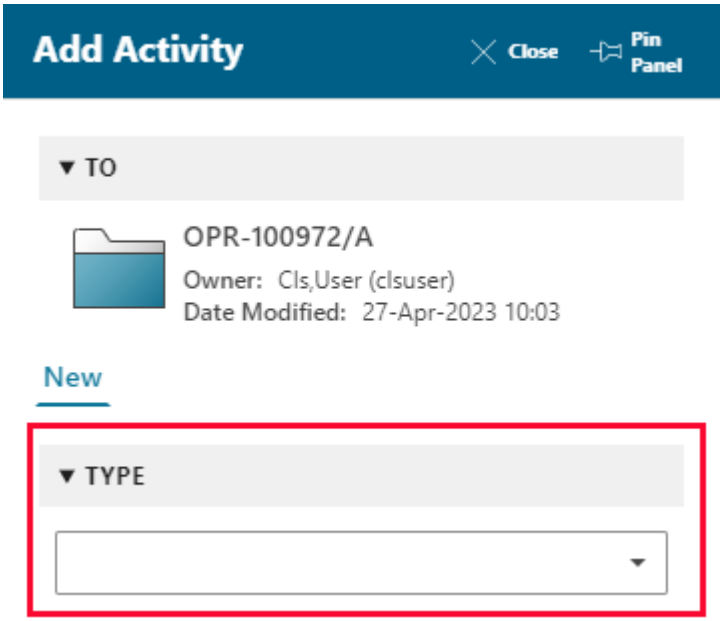
The **Add Activity** dialog box opens.



4. In the **Add Activity** dialog box, select the activity type from the **TYPE** list and enter the properties.

Name	Type the name of the new activity. It is recommended that you make the name unique so users can easily search the database using this term.
Description	(Optional) Type a description of the new activity.

5. Click **Add**.

   A message box appears confirming the activity is created.

Delete an activity

Delete an activity from the **Activities** panel on the **Part Manufacturing** tab.

1. Select the **Part Manufacturing** tab.
2. On the **Activities** panel, in the **Object** column, select a parent or child activity.

   > Note:
   > When you delete a parent activity, the child activities in the activity are also deleted.

3. Select **More commands** •••→**Edit** →**Delete** on the primary toolbar.

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4. In the confirmation message box, click **Delete**.

   The activity is removed from the **Activities** panel structure and from the database.

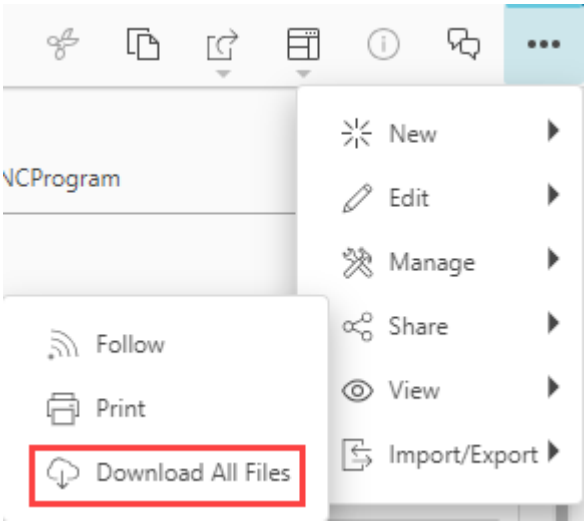
Download files from activities

Download a single file from an activity

Download a single activity's file from the **Activities** panel in **Part Manufacturing**.

Procedure

1. Select the **Part Manufacturing** tab.
2. In the **Activities** panel, expand the activities group that contains the activity file you want to download.
3. Select the activity that contains the file to download.
4. Select **More commands** •••→**Share** →**Download All Files** on the primary toolbar.



   A message is displayed confirming the number of files downloaded.

5. Click **OK**.

Download all files from an activity

Download multiple activity's files from the **Activities** panel in **Part Manufacturing**.

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Procedure

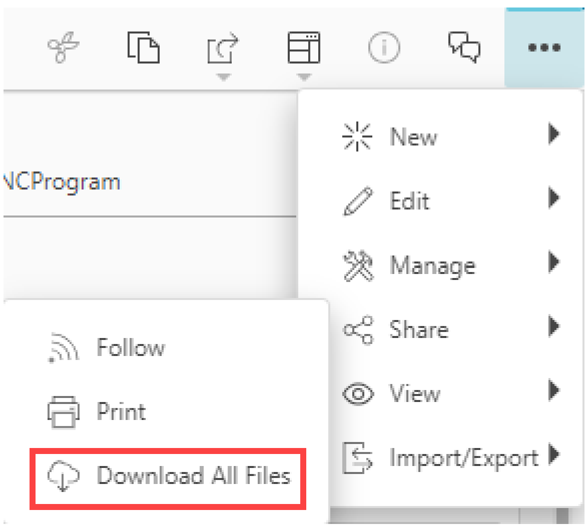
1. Select the **Part Manufacturing** tab.

2. In the **Activities** panel, expand the activities group that contains the activity files you want to download.

3. Select the activity that contains the files to download.

   Hold Ctrl and use the left mouse button to select multiple activities.

4. Select **More commands** •••→**Share** →**Download All Files** on the primary toolbar.

   A message is displayed confirming the number of files downloaded.

5. Click **OK**.

Download all files from an activity group

Download all files in the activity group from the **Activities** panel in **Part Manufacturing**.

Procedure

1. Select the **Part Manufacturing** tab.

2. In the **Activities** panel, select the activities group.

3. Select **More commands** •••→**Share** →**Download All Files** on the primary toolbar.

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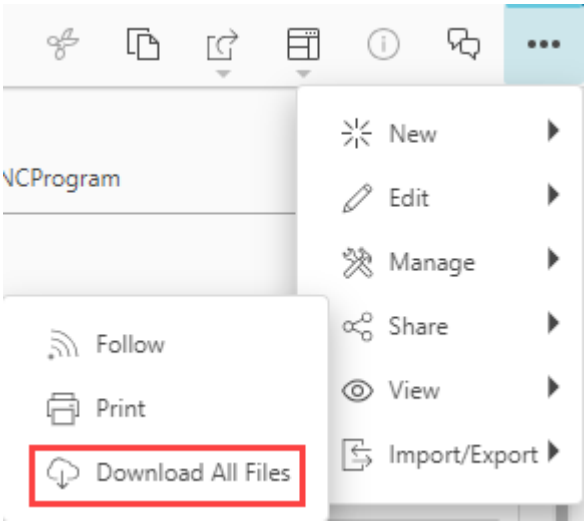

A message is displayed confirming the number of files downloaded.

All files in the activities group are downloaded.

4. Click **OK**.

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