Advanced Assemblies

Student Guide
May 2006
MT10230 — NX3

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Manual History

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	Version 16.0	January 2000
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A	NX 3	May 2006

This edition obsoletes all previous editions.

Contents

Course Overview

The Advanced Assemblies course will show you how to reduce the time required to open, manipulate and evaluate large assemblies. This will be accomplished by using simplified representations and by opening only designated portions of the assembly.

Intended Audience

The Advanced Assemblies course is intended for users who have been using NX assemblies and are ready to expand their skills and make use of the functionality provided by the Advanced Assemblies Module.

Prerequisites

This is an advanced level course. You should have taken the Design Applications using NX (DAU) course or have equivalent experience prior to taking this course.

Students entering this class should understand and be able to perform the following:

- Log on, navigate through a directory structure, create, open, close, and save part files.
- Open the Assembly Navigator and the pop up action windows.
- Create an assembly for Master Model implementation.
- Create and make use of reference sets.
- Assign and list user defined attributes.
- Change work part and displayed part.
- Utilize, understand and set Load Options

Course Objectives

After successfully completing the Advanced Assemblies course, you should be able to:

- Make use of Representations to quickly retrieve and manipulate large assemblies.
- Create and use Filters with user defined Zones and Attributes.
- Perform a Clearance Analysis and interpret the results.
- Calculate and manage the mass properties of an assembly using Assembly Weight Management.

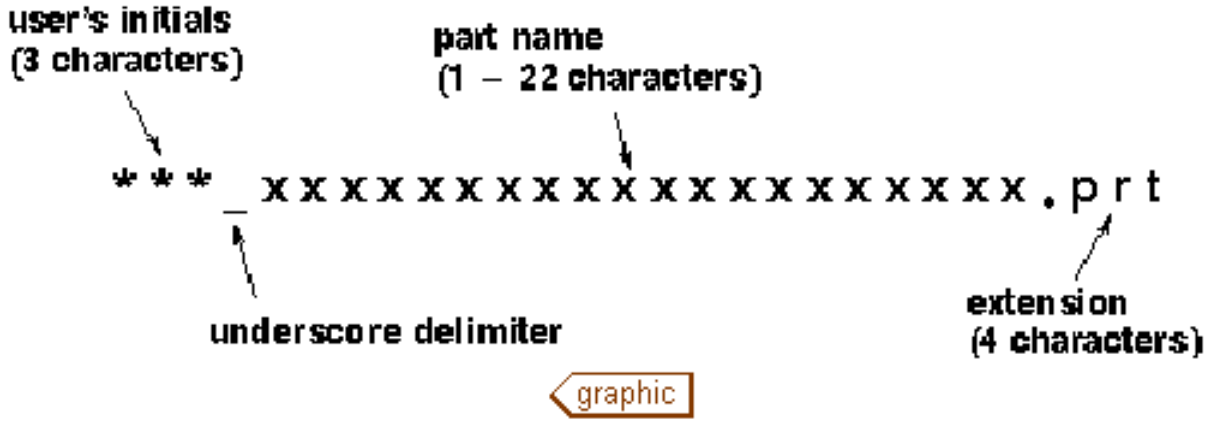
Classroom Part Files

You do not have the system privilege to modify any of the part files provided with this course. If you attempt to do so, you will get a system message saying that the file is Read Only and that you cannot save the changes. However, this does not restrict you from working on these files.

You can use the File→SaveAs option to save a copy of the current part file under a different name. This allows you to modify the part file and save the changes, if desired.

Class Part File Naming

This class utilizes the following file naming standard:



Where the student is requested to save a part file for later use, the initials of the student's given name, middle name, and surname replace the course identifier "***" in the new filename with the remainder of the filename matching the original. These files should reside in the "work" directory.

Up to 128 characters are valid for file names. A four character extension (.prt, for example) is automatically added to define the file type. This means the maximum number of user defined characters for the file name is actually 124.

Layers and Categories

The student will notice that standard layer assignments (as well as standard category names) have been used in the part files for this course:

Layers 1-100, Model Geometry (Category: MODEL)

Layers 1-14, Solid Geometry (Category: SOLIDS)

Layers 15-20, Linked Objects (Category: LINKED OBJECTS)

Layers 21-40, Sketch Geometry (Category: SKETCHES)

Layers 41-60, Curve Geometry (Category: CURVES)

Layers 61-80, Reference Geometry (Category: DATUMS)

Layers 81-100, Sheet Bodies (Category: SHEETS)

Layers 101 - 120, Drafting Objects (Category: DRAFT)

Layers 101 - 110, Drawing Borders (Category: FORMATS)

Layers 121 - 130, Mechanism Tools (Category: MECH)

Layers 131 - 150, Finite Element Meshes and Engr. Tools (Category: CAE)

Layers 151 - 180, Manufacturing (Category: MFG)

Layers 181 - 190, Quality Tools (Category: QA)

Colors

The following colors are the ones normally used to indicate different object types.

Object	Valid colors
Solid Bodies	Light Gray (87)
Sheet Bodies	Light Dull Azure (92)
Sketch Curves	Dark Hard Blue (212)
Datum Features	Light Weak Red (81)
Points, Saved Coordinate Systems, Lines,Arcs, Conics, Splines	Dark Hard Blue (212)
System Display Color	Red (4)

Seed Part

Seed parts are an effective tool for establishing customer defaults or any settings that are part-dependent (saved with the part file). This may include non-geometric data such as:

- Object Display Preferences
- Sketch preferences
- Layer categories
- User-defined views and layouts
- Part attributes

Two seed part files are available for use in this course. These files incorporate the standards described above, and include the TFR-TRI view as the default view.

- drf_seedpart_inch (Inches)
- drf_seedpart_mm (Millimeters)

How to Use This Manual

It is important that you use the Student Guide in the sequence presented because later lessons assume you have learned concepts and techniques taught in an earlier lesson. If necessary, you can always refer to any previous activity where a method or technique was originally taught.

The format of the activities is consistent throughout this manual. Steps are labeled and specify what will be accomplished at any given point in the activity. Below each step are action boxes which emphasize the individual actions that must be taken to accomplish the step. A toolbar icon may also appear at the end of the action text indicating that you may perform the same command from a toolbar icon. As your knowledge of NX increases, the action boxes will seem redundant as the step text becomes all that is needed to accomplish a given task.

Step 1: Open the part file **mypart** (the **prt** extension is always assumed).

☐ Select **File→Open**. 

While working through lesson activities, you will experience a higher degree of comprehension if you read the CUE and Status lines.

It is recommended that students who prefer more detail from an Instructor Led Course ask questions, confirm with restatement, and, more importantly, attend and pay attention to the instruction as it is given.

Obviously, it is always necessary for students to consider the classroom situation and be considerate of other students who may have greater or lesser needs for instruction. Instructors cannot possibly meet the exact needs of every student.

At the start of each class day you will be expected to log onto your terminal and start NX, being ready to follow the instructor's curriculum. At the end of the day's class you should always exit NX and log off the terminal.

Classroom System Information

Your instructor will provide you with the following items for working in the classroom:

Student Login:

Username: _____

Password: _____

Work Directory: _____

Parts Directory: _____

Instructor: _____

About This Training Environment

The login and working directories:

The work you will be doing will be performed in your login directory. These directories are named t01 through t16.

Your instructor will assign you a team number which will correspond to your login I.D. and password.

Your t# directory is where you will create your new files. The "car" directory is shared by all students. The component parts that you will be retrieving come from various sub-directories.

Lesson

1 Introduction to Advanced Assemblies

Purpose

To give you an overview of Advanced Assemblies.

Objectives

Upon completion of this lesson, you will be able to:

- Use Representations
- Use Filters
- Use the Assembly Navigator
- Perform a simple Clearance Analysis

1

Advanced Assembly / Large Assembly Issues

It is a commonly held belief that, generally, big jobs are made easier if they are broken into smaller and simpler tasks. That is what advanced assemblies is all about - allowing you to work on, or in the context of large assemblies without devoting time to retrieving and displaying data you do not need.

This is accomplished through the use of several tools including:

- Representations: Faceted objects associated to the parent solid. Representations can typically be retrieved and or shaded in a fraction of the time it would normally take to do the same with the solids. Representations can be used for display only or can be used to perform Clearance Analysis or Hidden Line Removal.

- Filters: Provide a way to select or work on parts that meet criteria set up by the user. Criteria can be based on spatial location (Zones), attribute values, Component Sets any combination of the three.

- Assembly Clearance: Lets you compare components to determine if they interfere or encroach on a minimum clearance distance that you define. This can be conducted interactively or in batch mode.

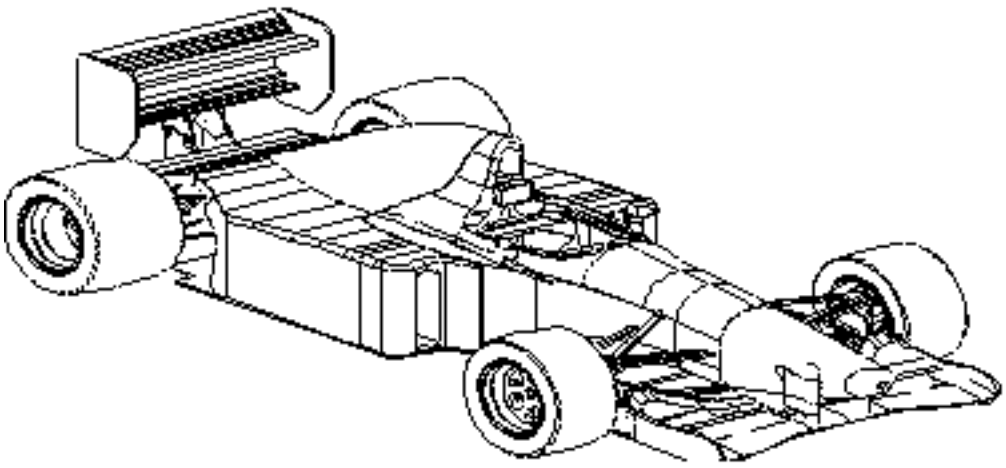
- Assembly Weight Management: Calculate and control the weight and other mass properties of parts, components, and assemblies based on their solids.

In this lesson you will work through an activity that will briefly introduce you to these tools. In later lessons you will learn about each of these tools in greater detail.

It is not the intent of this activity to teach you how to perform each of these functions, but rather to give you an overview of what is possible when using Advanced Assemblies.

Activity: Race Car Design Change

After completion of this activity, you will be exposed to the usefulness of Advanced Assemblies. You will get a hands-on introduction to the tools previously described. This activity does not show you how to set up your assemblies but illustrates the power of using Advanced Assemblies once parameters are properly set up. An explanation of how to set-up assembly parameters will follow.



Benetton Race Car

1

Set Up / Design Intent:

During this activity we will simulate the actions of a designer who needs to perform design changes to our "Benetton Race Car".

Required Changes:

Replace the current rear spoiler for a newer model.

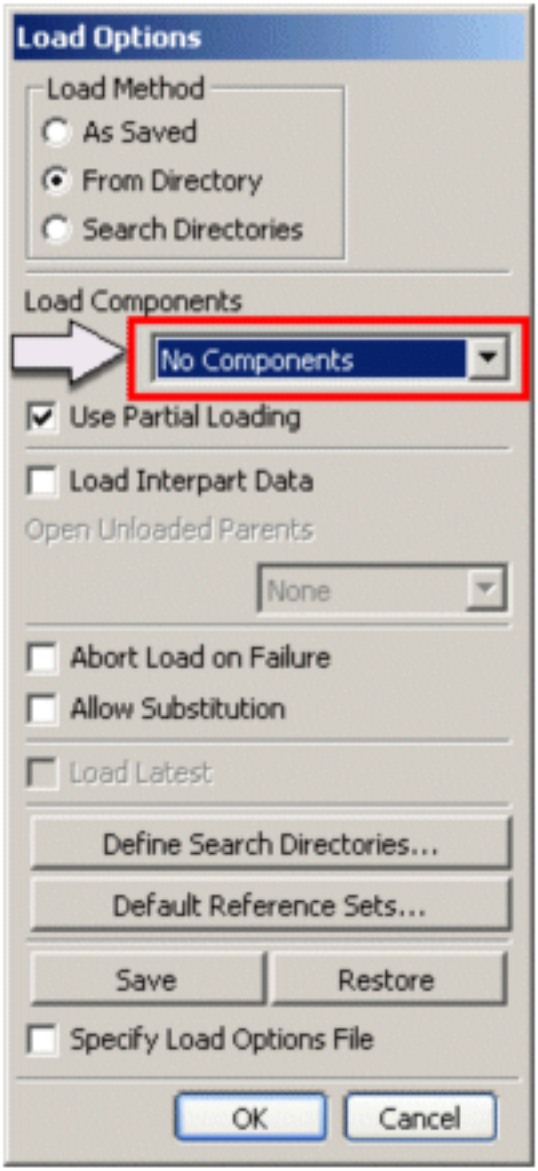
Move holes on the cyan mounting bracket to reposition the spoiler forward.

Perform a clearance analysis to determine acceptability of design changes.

Step 1: Set your load options to load No Components.

☐ Choose File→Options→ Load Options.

☐ Change the button under Load Components to No Components.



☐ Choose OK

Step 2: Open the part demo_mockup.prt from the car directory.

Notice this assembly retrieves very rapidly. This is because the assembly has had Representations created for it. The Representations will display when the solid is not loaded or is excluded from the reference set, that is why we set our Load Components option to the No Components setting.

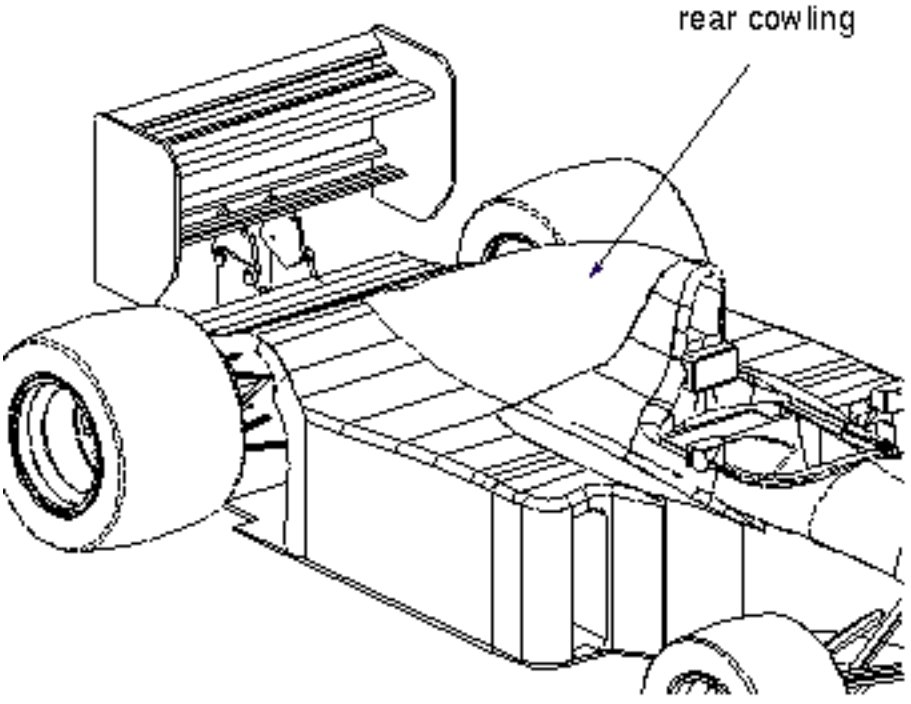
To see the increase in speed for other operations, you can quickly shade or perform a hidden line removal on the assembly.

Step 3: Perform a quick shade or a high quality shade.

☐ Choose the Shaded icon or MB3→Display→Mode→Shaded

To see more detail of the car, you could blank the representations for the rear cowling (see figure below).

Step 4: Choose Edit→Blank→Blank or the Ctrl-B keys. Select the cowling.



☐ Choose OK.

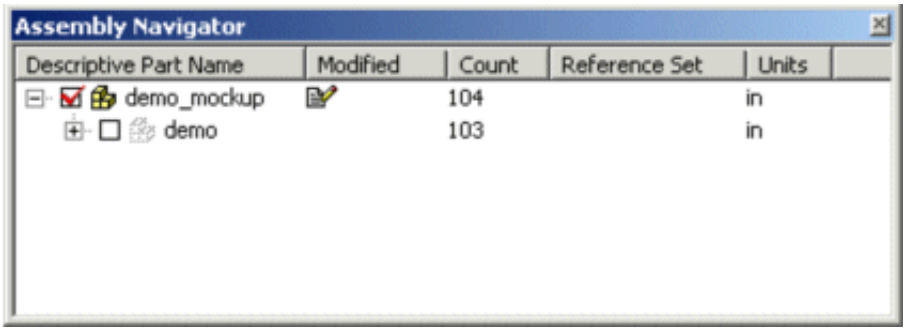
Zoom in on the back half of the car to see the engine compartment.

Step 5: Open the Assembly Navigator and size to your liking.

☐ Choose the Assemblies icon from the Application toolbar or choose Application→Assemblies.

☐ Choose the Assembly Navigator tab on the Resource Bar

(Unix users, choose the Assembly Navigator icon on the Navigators toolbar.)



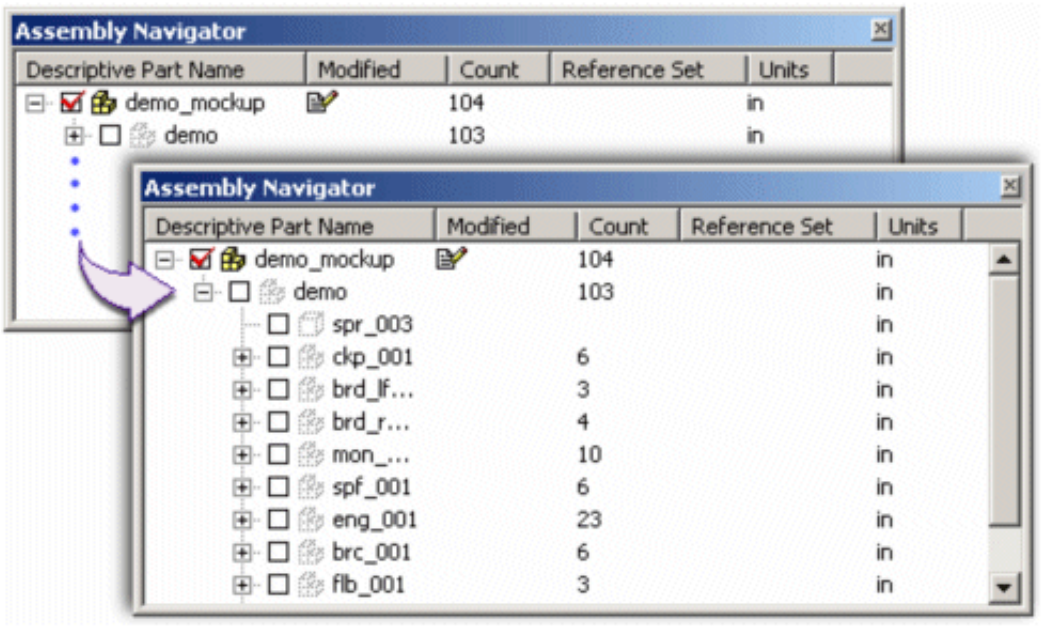
This is the Assembly Navigator. You have access to all the components via the Assembly Navigator even though they are not loaded nor currently shown. Notice the Master Model technique was used to build this part file. The part "demo" was added to "demo_mockup". This was done because many of the advanced assemblies "objects" are stored in the top assembly.

Step 6: Change your display mode back to wireframe.

☐ Choose the Wireframe icon.

Step 7: Highlight the Bounding Boxes.

☐ Click MB1 on the plus sign in front of the demo component to expand its component list.



Move your cursor over any Assembly Navigator node (any component listed in the Assembly Navigator is a node). Notice that

sub-assemblies do not have bounding boxes (ie. ckp_001). Only components containing solids show the bounding box. (ie. spr_003).

☐ Choose the Expand All icon and repeat the previous action.

A red box will highlight in the graphics screen indicating the location of the component and the "minimum volume" surrounding that component. This minimum volume or "Bounding Box" is important and will be discussed later. You can use either of the two previous actions to expand part or all of the component lists. Next you will perform some design changes in the context of the assembly. You will replace and move the rear spoiler as a result of wind tunnel testing. Engineering analysis indicates the spoiler should move forward and change shape. You will also use Bookmarks and Filters to find and load all the components located near the cyan mounting bracket in order to change the context of the assembly. Then you will perform a Clearance Analysis to determine if the changes have introduced any interference problems.

Step 8: Find the component name of the cyan mounting bracket for the rear spoiler, then load that component. To focus your search, we will need to change your Load Options.

☐ Choose File→Options→ Load Options.

☐ Change your Load Method to Search Directories .(1)

☐ Choose Define Search Directories .(2)

☐ Select Session Dirs.(3) This will add the directory the car was loaded from to your list.

☐ Select the directory from the Search Directories list. (4)

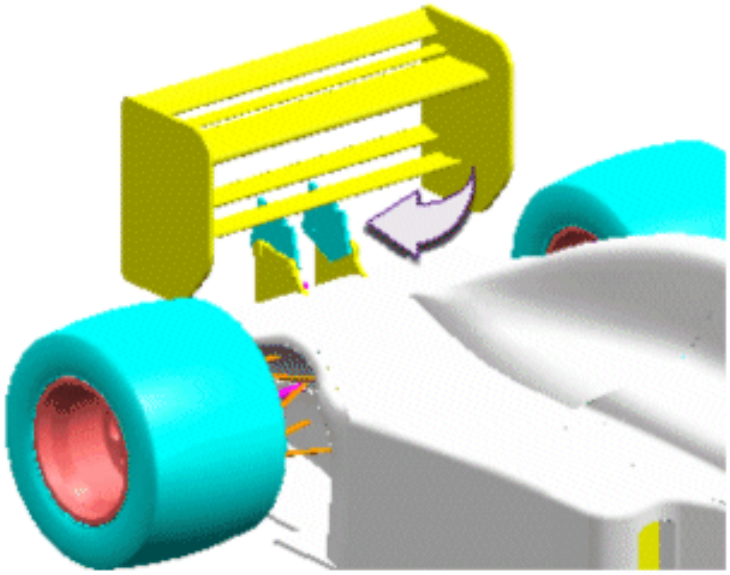
☐ Edit the New Directory entry by adding "..." to the end, then Enter. (5)

☐ Choose OK. (6)

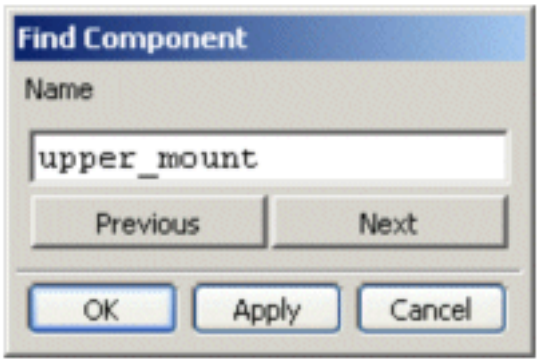
☐ Choose the Find Selected Components icon on the Assembly Navigator toolbar or choose Tools→Assembly Navigator→Find Selected Components.

☐ Select the cyan mounting bracket from the graphics screen.

The representation will not pre-highlight.

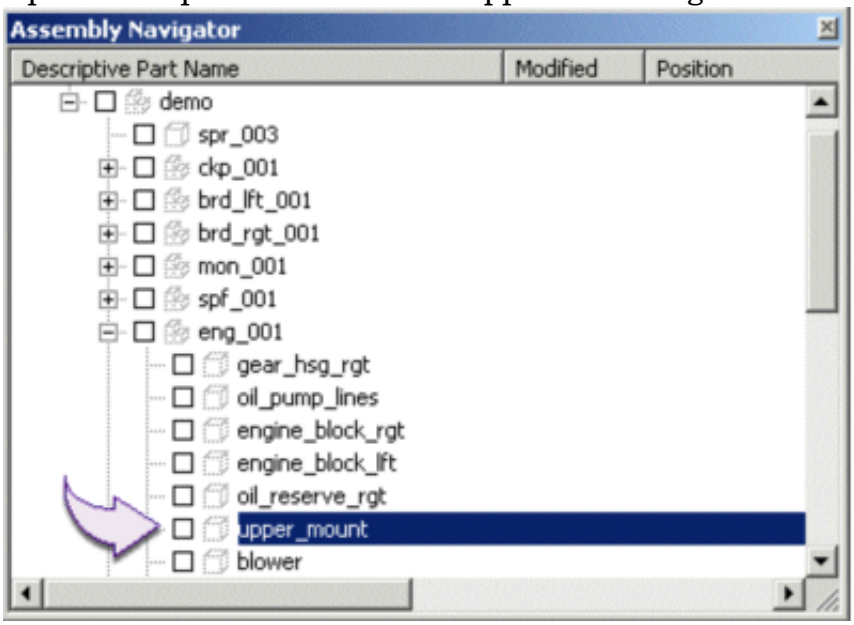


This will automatically fill in the name of the component in the Find Component dialog box and center the component within your Assembly Navigator.



☐ Choose OK

☐ You can click on the box in front of the upper_mount or, using MB3 on the upper_mount node, choose Open→Component to load the upper mounting bracket.

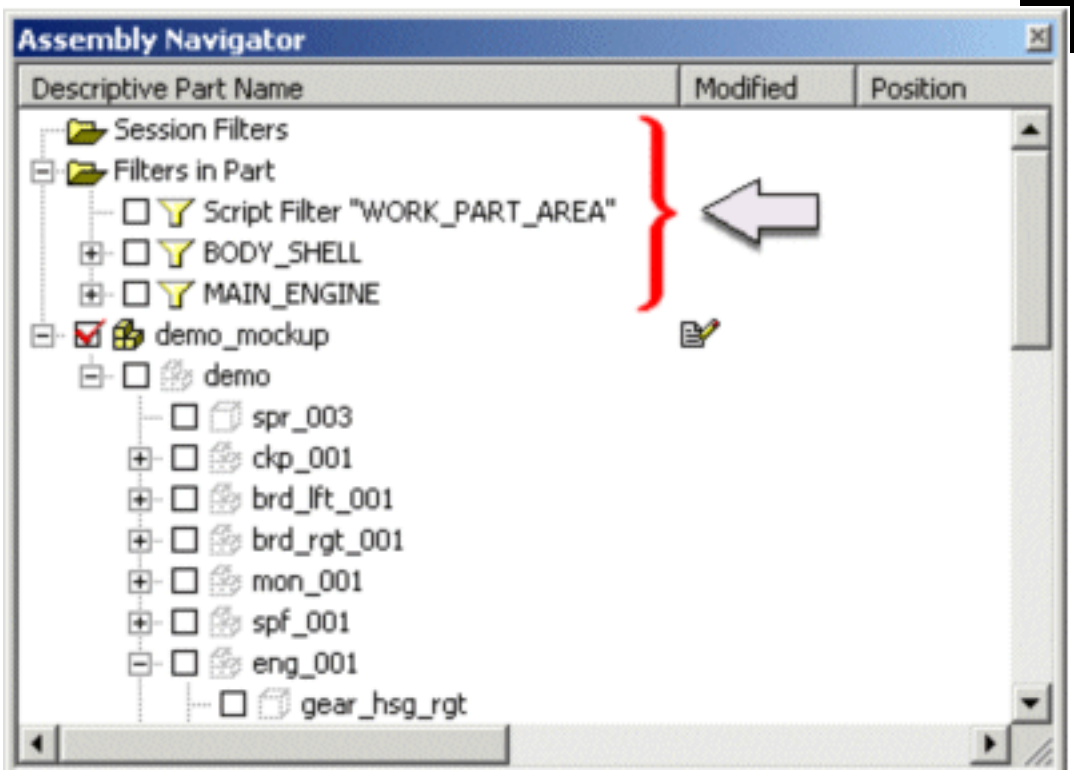


Now you are going to make use of Filters to load all the parts within 8 inches of your work part but not members of the engine component set or the body panels component set. A filter was created for you. Later, we will discuss the steps you are about to follow.

Step 9: Turn on the Filtering Mode.

☐ Choose the Filtering Mode icon on the Assembly Navigator toolbar or choose Tools→Assembly Navigator→Filtering Mode.

Your Assembly Navigator should now show the filters at the top.



☐ Change your work part to the upper_mount.

Step 10: Retrieve a previously created filter.

☐ Choose the Open icon on the Standard toolbar or choose File→Open.

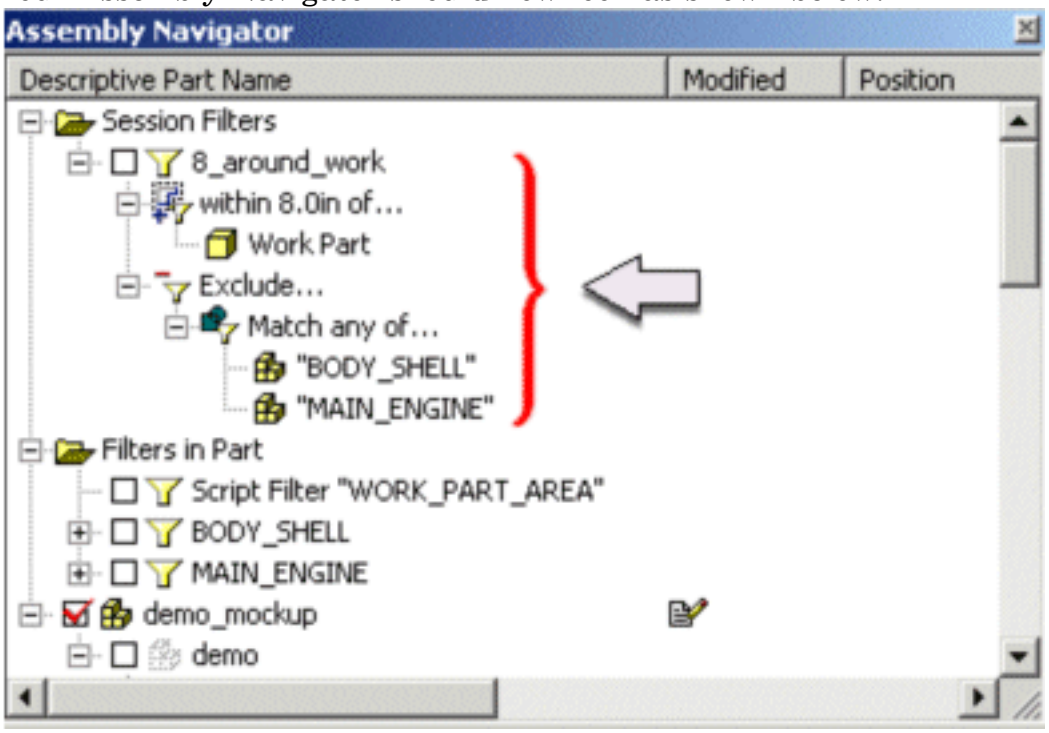
☐ Change the Files of Type option to Bookmarks.

☐ In the Files window, choose work_part_area.bkm.

☐ Choose OK.

☐ Click MB1 on the 8_around_work plus sign (+) and its expanded components to show all qualities of the filter.

Your Assembly Navigator should now look as shown below:



This filter can be read as retrieving all parts within 8 inches of the workpart volume excluding the BODY_SHELL component set and excluding the MAIN_ENGINE component set.

☐ Click MB1 on the box for the 8_around_work filter.

Your status line should report to you that it is loading several parts. Notice the parts that are loading are only the ones in the immediate vicinity of your work part but do not include any of the engine parts or body parts.

There are a couple of methods you can use to give you visual cues as to which components are loaded, showing their solid geometry, and which are not loaded but showing the representations.

Step 11: Here are two ways to determine if you are viewing a solid or a representation.

☐ Make demo your displayed part.

Now, since the representations are stored in another part file, demo_mockup, the representations are no longer visible and the only parts you see are loaded components displaying the solid model.

☐ Change your displayed part back to demo_mockup.

☐ Enable partial shading by accessing the MB3 pop up menu from the graphics window and choosing Display Mode → Partially Shaded

This is only available when you have hardware shading available. The effect is that your loaded solid models will appear shaded and the representations will be shown in the normal wireframe display.

Step 12: Now make changes to the mounting brackets to bring the spoiler forward.

☐ Change your work part to upper_mount if it is not already there.

☐ With the upper_mount as your work part, change your application to Modeling.

☐ Choose Tools→Expression.

☐ Select the expression forward_hole_offset, change its value to 1.5, then Enter.

☐ Choose OK.

Since you have mating conditions in place, the spoiler will move to the new desired position automatically.

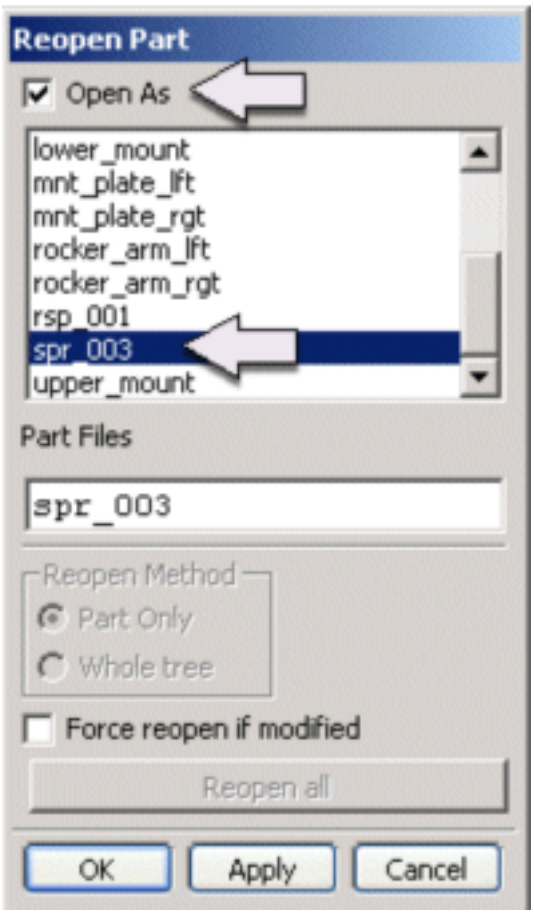
Step 13: Now replace the current spoiler with a newer model. The trick here is to put in a new spoiler without losing any mating conditions.

☐ Change your work part back to demo_mockup.

☐ Select File→Close→Reopen Selected Parts.

☐ Toggle the Open As switch on.

☐ Select spr_003 from the component selection list.



☐ Choose OK

☐ Select spr_003_new from the spoiler_rear subdirectory.

☐ Choose OK.

An Information window will appear verifying your changes.

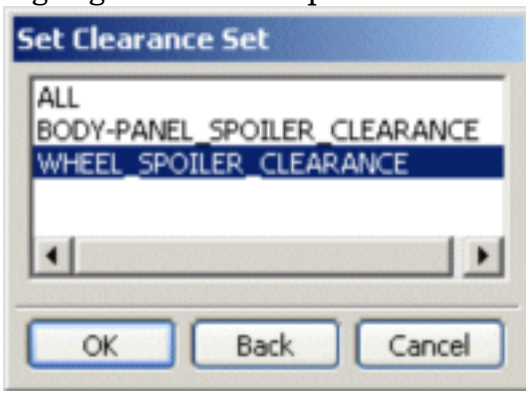
☐ Dismiss the Information window

Step 14: You will now perform a Clearance Analysis to determine if the new spoiler interferes with any other parts or encroaches on their minimum distance requirements. The engineer determined that 4 inches clearance is necessary to prevent the wheels from rubbing the spoiler. That requirement was set up in the Clear Zones dialog under the title Wheel_Spoiler_Clearance.

☐ In the Assembly Navigator, click on the box in front of both rear_tire nodes to load them.

☐ Choose Analysis→Assembly Clearance→Clearance Set→Set .

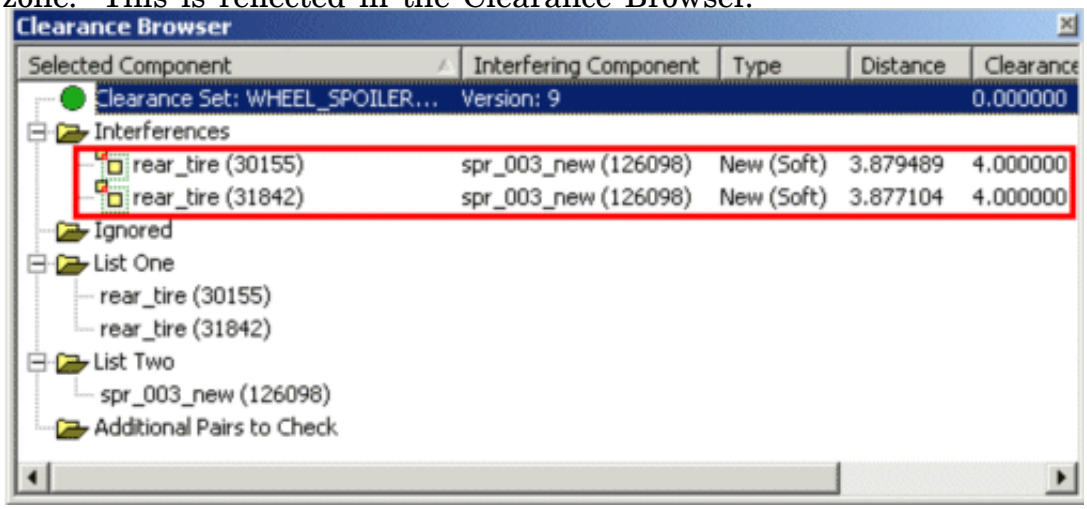
☐ Make sure the Wheel_Spoiler_Clearance clearance set is highlighted in the top window of the dialog, then choose OK.



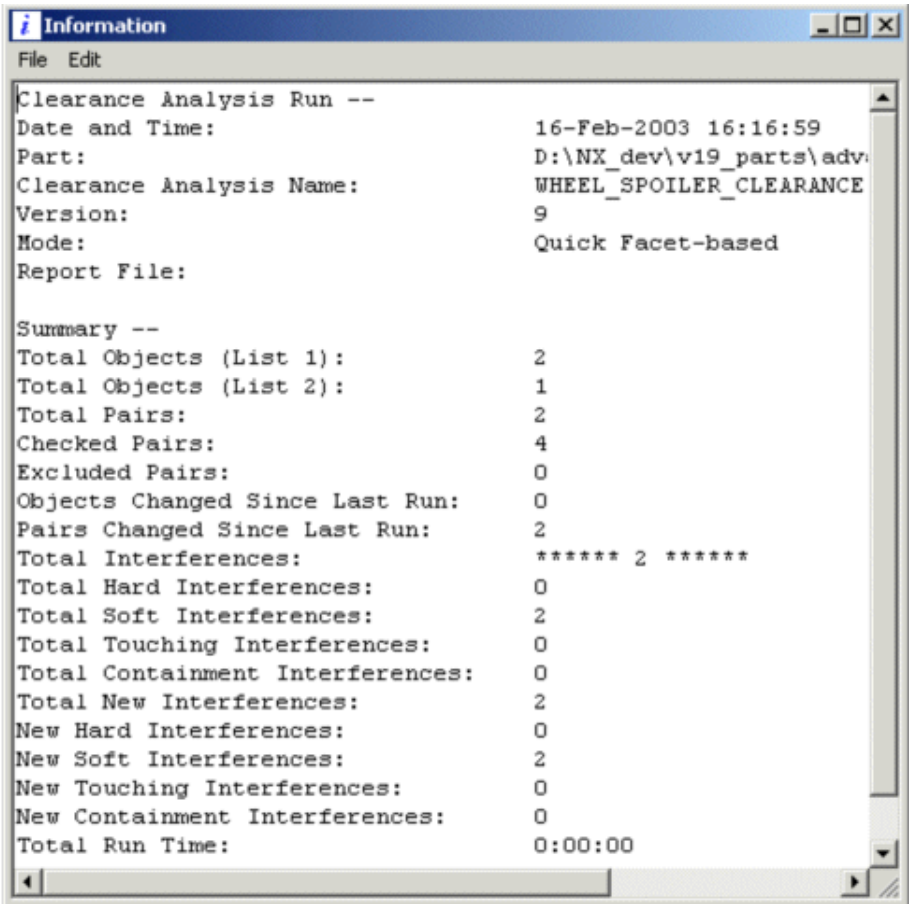
☐ The Clearance Browser appears showing the Wheel_Spoiler_Clearance Clearance Set.

☐ Select the Clearance Set in the Clearance Browser, then use Perform→Analysis.

The system will then proceed to do the clearance analysis. It should find two soft interferences, in other words, it should find two areas violating the clearance zone. This is reflected in the Clearance Browser.



☐ With the Clearance Set node still selected in the Clearance Browser use MB3→Summary.



Step 15: Close your part

Summary

In this lesson you:

- Loaded assembly representations.
- Loaded parts within eight inches of the workpart excluding parts from the "engine" or "bodypanel" (so mates update).
- Changed a design in context.
- Replaced the new spoiler for the original.
- Checked for interferences as a result of your edits.

1

Lesson

2 Component Filters

Purpose

To learn how to manipulate assemblies using Bookmarks, Attributes, Filters and Zones.

Objectives

Upon completion of this lesson, you will be able to:

- Use Bookmarks, Filters and Zones.
- Create Filters & Bookmarks and use Zones to manipulate your assemblies.
- Identify the differences between the new Component Filtering interface and the pre-V16 component filtering interface.

Introduction to Component Filters

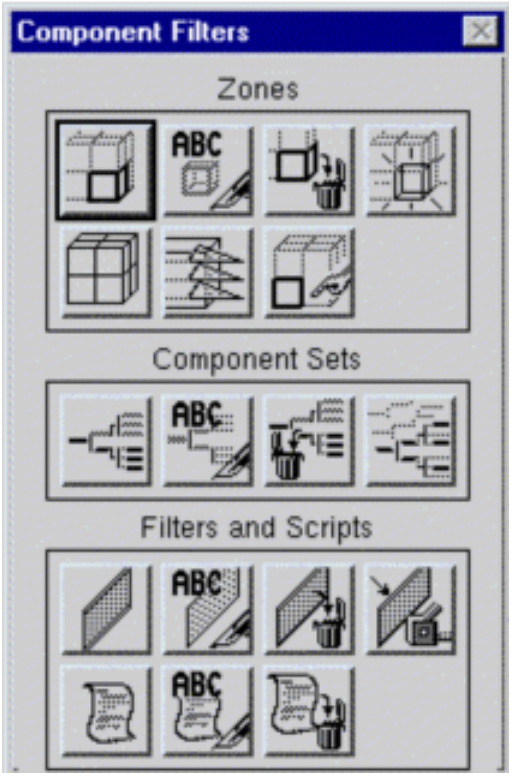
2

NX provides a way for you to manipulate assemblies based on predetermined criteria. The criteria available for use includes:

- Attributes
- Components Lists
- Zones and spatial location

In this lesson you will work with several assemblies using each of the different methods of manipulation.

It is important to understand just what component filters are and what they are used for. A component filter, just like a conventional physical filter, is used to filter or strain unwanted or unneeded things out of your assemblies. Where a physical filter uses size as the criteria for acceptance or non-acceptance, component filters make use of Zones and Attributes as the comparison criteria.



Only portions of the pre-V16 filtering interface will be discussed in this manual. The current interface is a replacement for the pre-V16 interface. The pre-V16 interface is still available, but will not be discussed much.

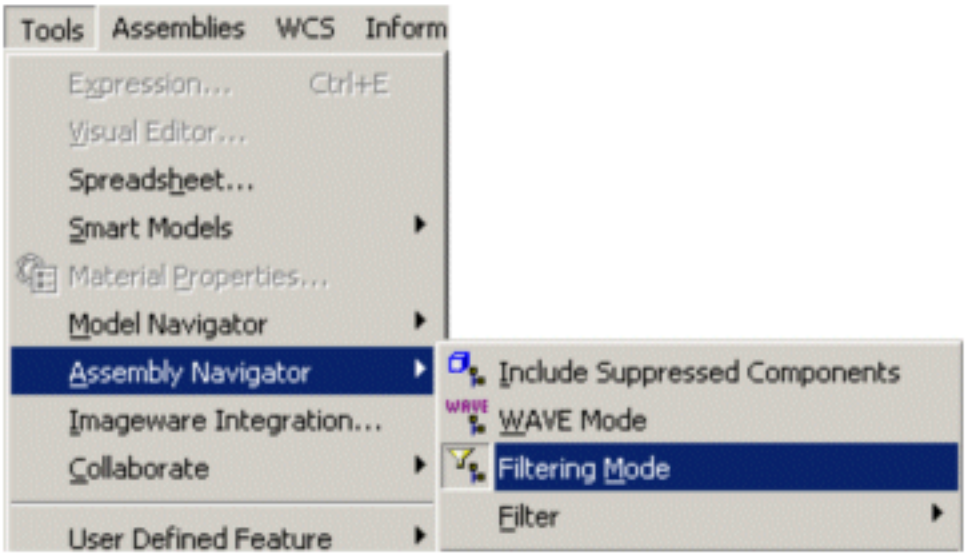
Assembly Navigator Filters

Filters are invoked from the Assembly Navigator popup menu or Filtering toolbar. Filters let you conditionally apply actions to all or part of the assembly structure.

Filters let you navigate an assembly and locate parts that interest you, without opening the entire assembly. Filters can locate parts based on spatial properties (such as proximity to a specified component) or based on part attributes and properties.

Filtering, by itself, does not differentiate between different solids in a part file. If you need to do this, you should make use of reference sets.

In order to see and use filters in the Assembly Navigator, the Filtering Mode must be turned ON. You can turn it ON either by choosing Tools→Assembly Navigator→Filtering Mode. The Filtering Mode has its own toolbar.



Assembly Navigator Toolbar:

- Include Suppressed Components: suppressed components within an assembly are included in the Assembly Navigator tree
- WAVE Mode: Toggles the setting for the Assembly Navigator between the standard Assembly Navigator and the WAVE Assembly Navigator (which has additional popup options).
- Filtering Mode: When this option is toggled ON, the Assembly Navigator displays all the defined filters for the current assembly.

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-  Find Selected Components: Lets you find occurrences of a specified component.
-  Find Work Part: Selects and highlights the current work part in the Assembly Navigator and the graphics window. This option causes the Assembly Navigator to scroll, if necessary.
-  Collapse All: Collapses the Assembly Navigator part nodes so that only the first level components are shown. (The root node is not collapsed.)
-  Expand All: Expands all collapsed nodes in the Assembly Navigator so that every component has a visible node.
-  Expand to Selected: Expands the Assembly Navigator to show the selected components after components have been selected by means other than the Assembly Navigator.
-  Expand to Visible: Expands all collapsed nodes that contain visible components.
-  Expand to Work: Expands collapsed nodes that contain the work part. The Assembly Navigator scrolls, if necessary.
-  Expand to Loaded: Expands all collapsed nodes that contain fully and/or partially loaded parts.
-  Pack All: Packs all components that have multiple occurrences in the assembly. If you add another occurrence later, it is not automatically packed.
-  Unpack All: Unpacks all components that have multiple occurrences.
-  Export to Browser: Exports the contents of the Assembly Navigator, in HTML format, to an internet browser

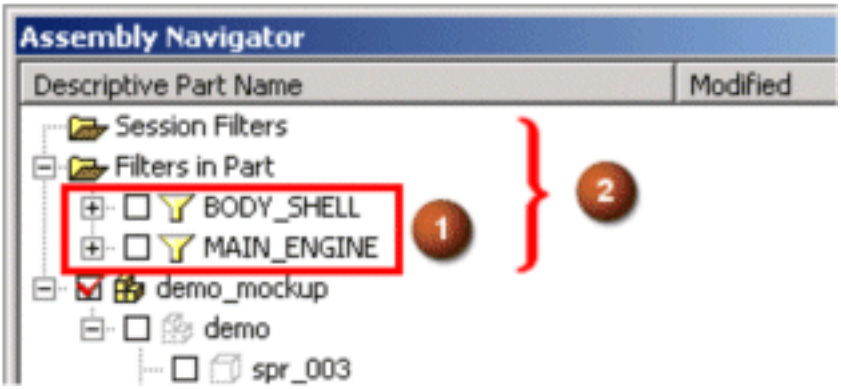
Filtering Toolbar icons:

-  Create Filter: Creates an empty filter, which you can then modify.
-  Add Attribute Search to Filter: Lets you add a Attribute Search to the selected filter. When no filter is selected, this creates a new search filter.
-  Add Proximity Filter: Adds a proximity condition around the selected filter. When no filter is selected, this creates a new filter.
-  Add Zone to Filter: Lets you add a zone to the selected filter. When no filter is selected, this creates a new filter.
-  Match all of... Filter: Combines the selected filters into a Match all of ... filter. When no filter is selected, this creates a new filter.
-  Match any of ... Filter: Inserts a Match any of ... filter around the selected filters. When no filter is selected, this creates a new filter.
-  Exclude from Filter: Lets you exclude selected components from the selected filter at that level.
-  Delete Filter: Deletes the selected filter(s) or portions of a filter.

2

Component Filters:

The filtering nodes are separated in the Assembly Navigator from the standard nodes. In the figure below, the Filtering Mode information appears at the top of the Assembly Navigator. It includes two categories: Session Filters (2) and Filters in Part (1). Also shown, are two Component Sets created prior to V16.



Filter Types

There are two categories of filters:

- Functional filters, which use assembly data:

  Add Attribute Search to Filter

  Add Proximity Filter

  Add Zone to Filter

- Organizing filters, which allow groupings and combinations of functional filters;

  Match all of ... Filter

  Match any of ... Filter

  Exclude from Filter

Terms and Definitions:

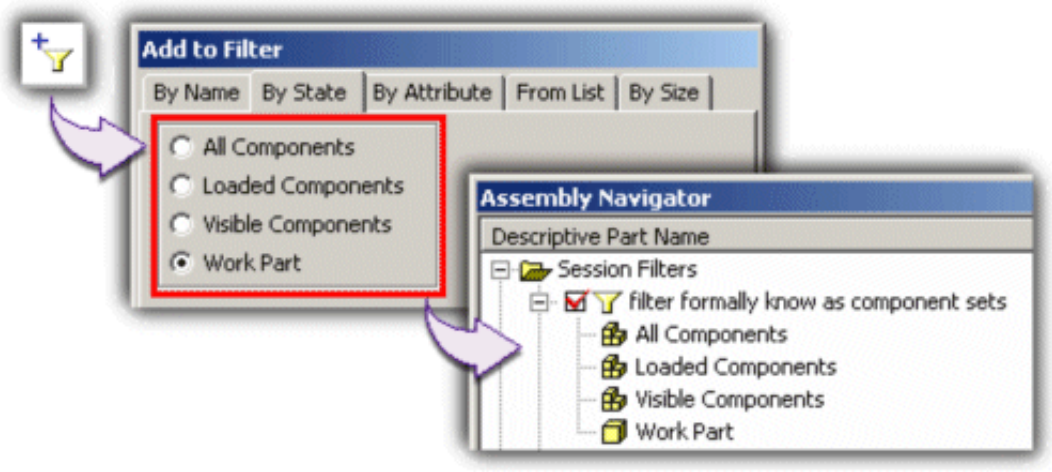
Component Set: A named list of components contained within an assembly (pre-V16 term).

Zone: A named region of space within your assembly model.

Attribute: Annotations that can be "attached" to components. Examples include color, layer, material, weight, cost, etc.

Filter: Analogous to the class selection dialog, identifies what components will be acted upon. Components to be acted upon are determined by use of the comparison criteria described above.

System Defined Component Sets



There are four system defined component sets. They are:

- All Components: The default. This state filter holds all components.
- Visible Components: Fully or partially loaded components that are visible.
- Loaded Components: All fully or partially loaded components visible or not visible.
- Work Part: The current work part.

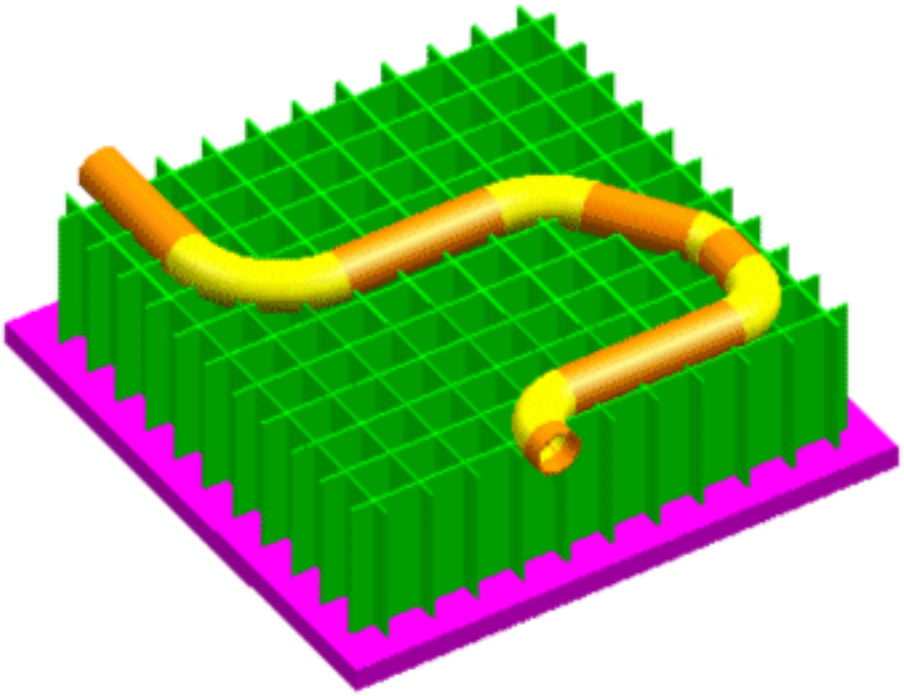
Any component set created through the pre-V16 user interface will also appear under the "Component Sets in Part" folder.

Activity: Filter Creation and Use

2

This activity will familiarize you with the creation and use of a filter used to select components. This is comparable to the Component Set function in the pre-V16 user interface.

This activity makes use of a simple assembly to bring your attention to the concepts presented.



Step 1: Open the assembly aa_cset.prt from the act1 directory.

☐ Set your Load Options to load All Components and Load Method to From Directory.

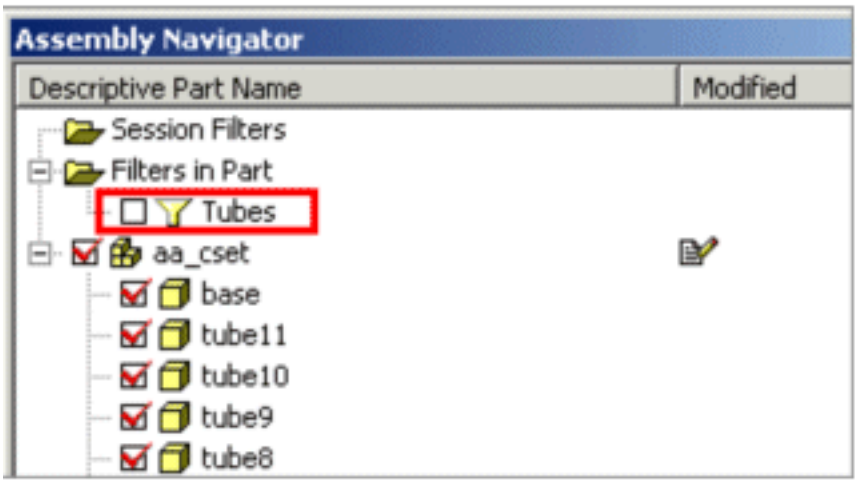
☐ Open the assembly aa_cset.prt from your act1 directory.

Step 2: Create a filter called Tubes which contains all tubes.

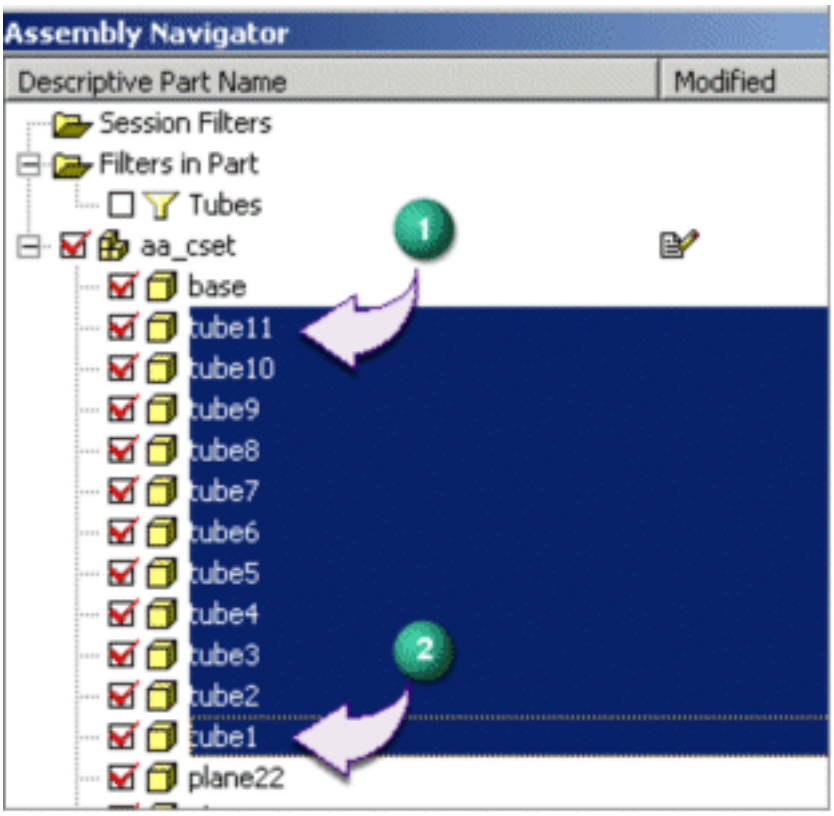
☐ Make sure the Assembly Navigator is opened and the Filtering Mode is on .

☐ Choose the Create New Filter icon.

☐ Enter the name *Tubes* for the new Filter



☐ Select all of the tube components from the Assembly Navigator (click MB1 on the top tube node, then shift-MB1 on the tube node at the bottom of the list).



☐ Using MB1 drag the tube components into the Tubes Filter

☐ Click once on the check box to show count (should be 11)

2

☐ Click again to blank and once more to unblank

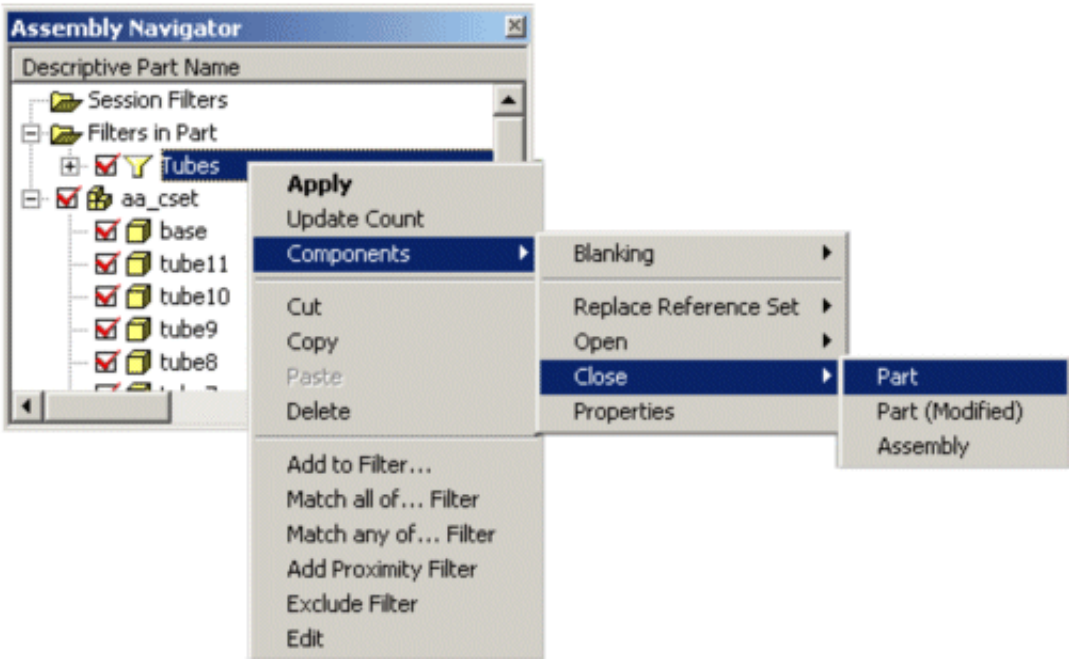
You now have a filter ready for use.

What if you wanted to load only the tubes? You can accomplish this in several ways. A simple method is to close the components in the assembly and then open the components in your filter.

Step 3: Move your pointer over the "aa_cset" assembly node, click MB3 and choose Select Assembly (all components highlight) then MB3 again and choose→Close→Assembly

Step 4: Click on the tubes filter check box. If it is checked, you will need to click it twice.

This demonstrated your ability to open and close components that belong to a specific filter

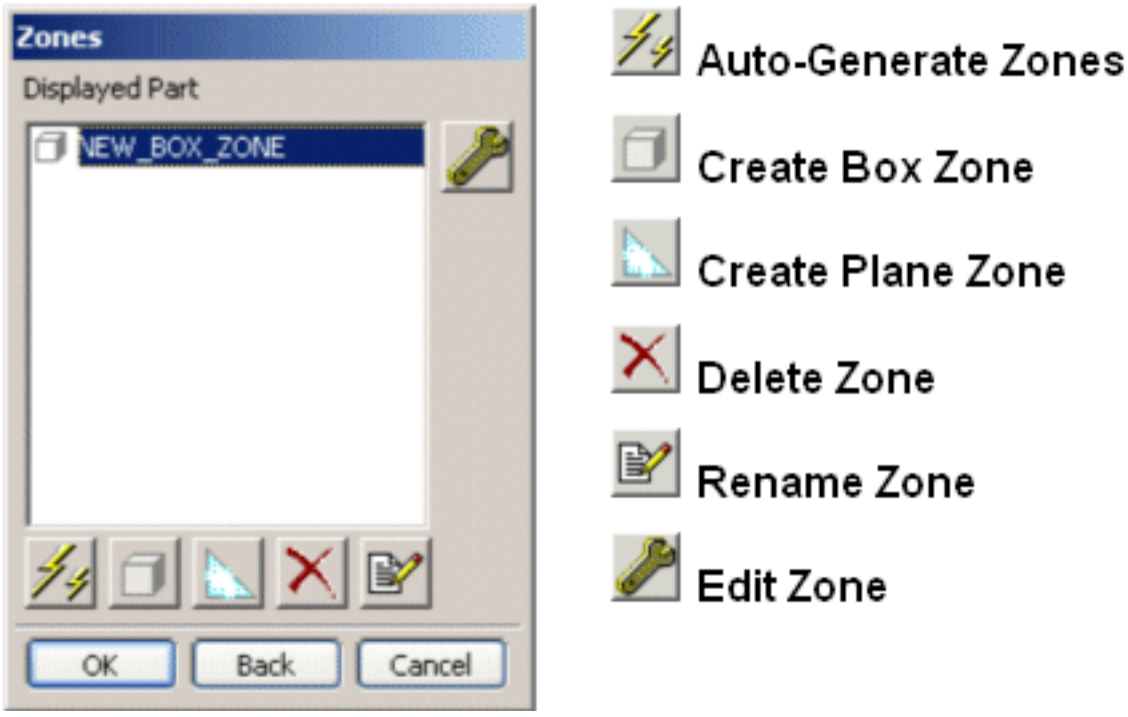


The component cascade has the most common actions performed on a component filter.

Step 5: Close your part file.

ZONE Definition and Use

NX allows you to define and name regions of space that can be used later as filter criteria. These regions of space are referred to as Zones. An assembly may be broken into several zones. A user could choose to open an assembly looking at one or more of the zones. This dialog box is found by choosing Assemblies→Component Filters.



NX gives you the option of either auto-generating zones for your entire assembly or explicitly sizing and placing a zone in the desired location.

Zones can be used within filters. An example might be, a filter containing components within a "box zone" and lie above a "plane zone".

NX determines if components intersect or cross a zone based on the "Bounding Box" that envelops the bodies within each component.

Bounding Boxes

A bounding box is defined as the minimum box shaped envelope that could contain your part. If a component is unloaded in an assembly, the system uses the bounding box for that component in consideration for zone inclusion. There are some important points that you should understand regarding bounding boxes.

- The bounding box size and orientation are determined by the absolute CSYS at the part level.

  If you have more than one solid in your part, the way the system defines your bounding box is defined as follows:

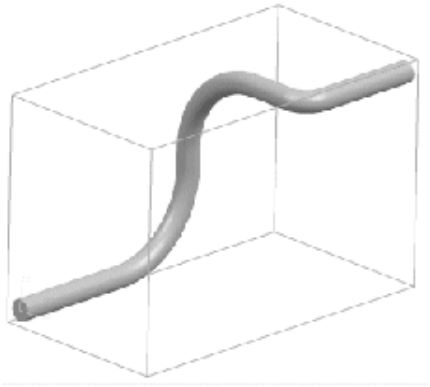
- The bounding box will be sized to accommodate all solids and or sheet bodies regardless of whether they are blanked or in your current reference set. OR

- You can specify a reference set to determine which bodies contribute to the overall bounding box of a part, by setting the customer default variable Assemblies_ModelReferenceSet to the name of that reference set. If that reference set does not exist in a part, all solids/sheets will be used. This is useful to exclude construction or tool solids from the bounding box calculation.

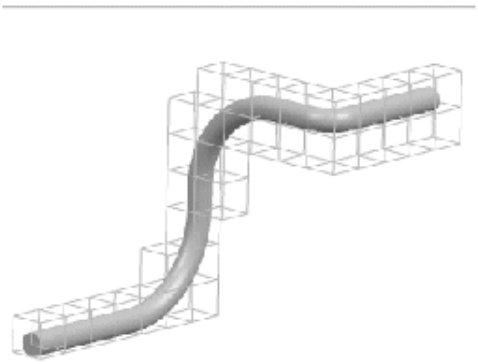
You can view the bounding box of an unloaded component by moving the pointer over its node in the Assembly Navigator or by using the Information→Object→Type→Component method. This is not true for sub-assemblies or wireframe parts.

True Shape Filtering

As discussed above regarding Bounding Boxes, an unloaded component's size and location are approximated by the bounding box definition for that part. Due to the simplified nature of bounding boxes this can drastically exaggerate the relative size of a component as shown in the figure below.



Bounding Box



True Shape Filtering

NX allows for a more refined "bounding box" definition using True Shape Filtering.

True Shape Filtering divides space into a grid of cubic cells. The size (or edge length) of these cubic cells is defined by the Assemblies_TrueShapeCellSize customer default. A components "refined bounding box" becomes the conglomeration of the small cells that contain a portion of the component.

True Shape filter evaluation can take longer than bounding box evaluation, but if the filter is being used to open components, using True Shape can save time overall by reducing the number of extraneous components that are loaded and opened. This is particularly true when dealing with tubes, wiring, cables, thin-walled parts like skins , etc.

If this functionality is desired to be used it is recommended that system administrators set the following customer defaults for their entire site. These defaults are needed to enable True Shape filtering:

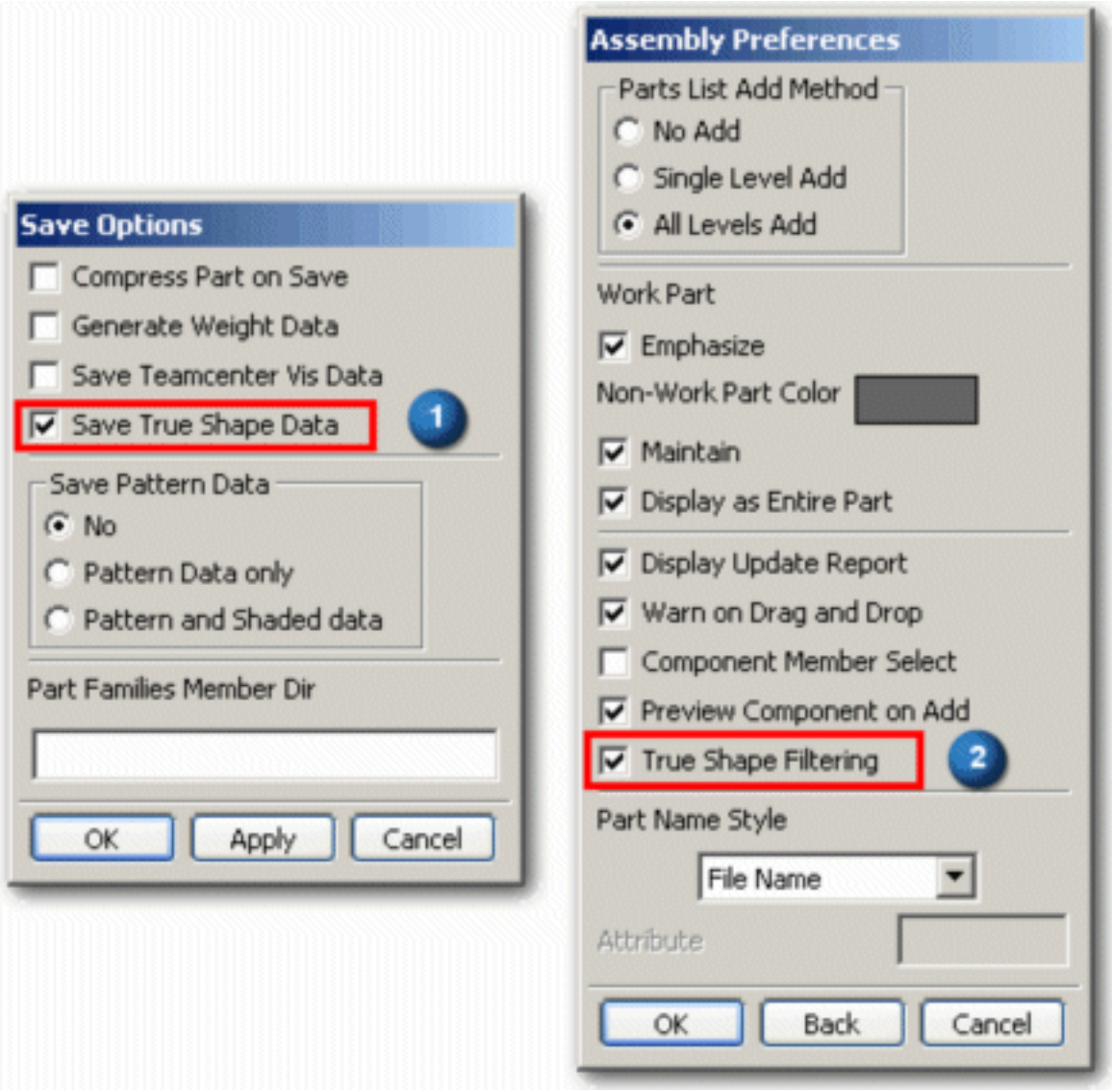
- Set the *Assemblies_TrueShapeGenerate* customer default to yes.
- Define a cell size for the Assemblies_TrueShapeCellSize customer default.
- Define the units for the cell size with the Assemblies_TrueShapeCellSizeUnits customer default.

2

Once this has been done, True Shape Filtering can be enabled by:

Toggling on the Save True Shape Data option on the Save Options dialog (File→Options→Save Options) (1) to allow occupancy data to be generated upon saving a component.

Toggle ON True Shape Filtering in the Assembly Preferences dialog (2).



True Shape Filtering can be toggled off if you wish to use regular bounding boxes.

Create Box Zone

This option allows you to create a box shaped zone. The following option are given as methods to define the size of the box zone.

- Edge Lengths, Corner
- Height, Two Points
- Two Diagonal Points

Once the Zone has been created you can use the Rename option to give it a logical name.

The box orientation will be relative to the WCS. Once the zone has been defined, the system will show a temporary display of the zone boundaries in the current system color.

Create Plane Zone

This option allows you to create a plane zone. The definition options are the same as you get with any plane subfunction dialog. Once the plane zone has been defined, the system will temporarily display the plane as well as its normal vector and name. The direction of the normal vector is important as will be seen shortly.

Edit Zone

If you select an existing box zone from the list, the system will allow you to Change Box Lengths or use Transform to translate, reposition, or rotate the zone.

If you select an existing plane zone, the system will allow you to use Transform to translate, reposition, or rotate the plane or use Flip Normal to reverse the direction of the normal vector.

2

Rename Zone

- Allows you to rename an existing zone. Zone name rules are:
- 29 characters maximum.
- No embedded spaces allowed.
- Not case sensitive, i.e. whether you enter upper or lower case letters the system will convert your input to upper case.
- Valid leading characters are limited to, _, and -.
- All zones must have unique names

If you wish to view a zone that you have created just select the zone from your list and it will temporarily highlight in your current system color.

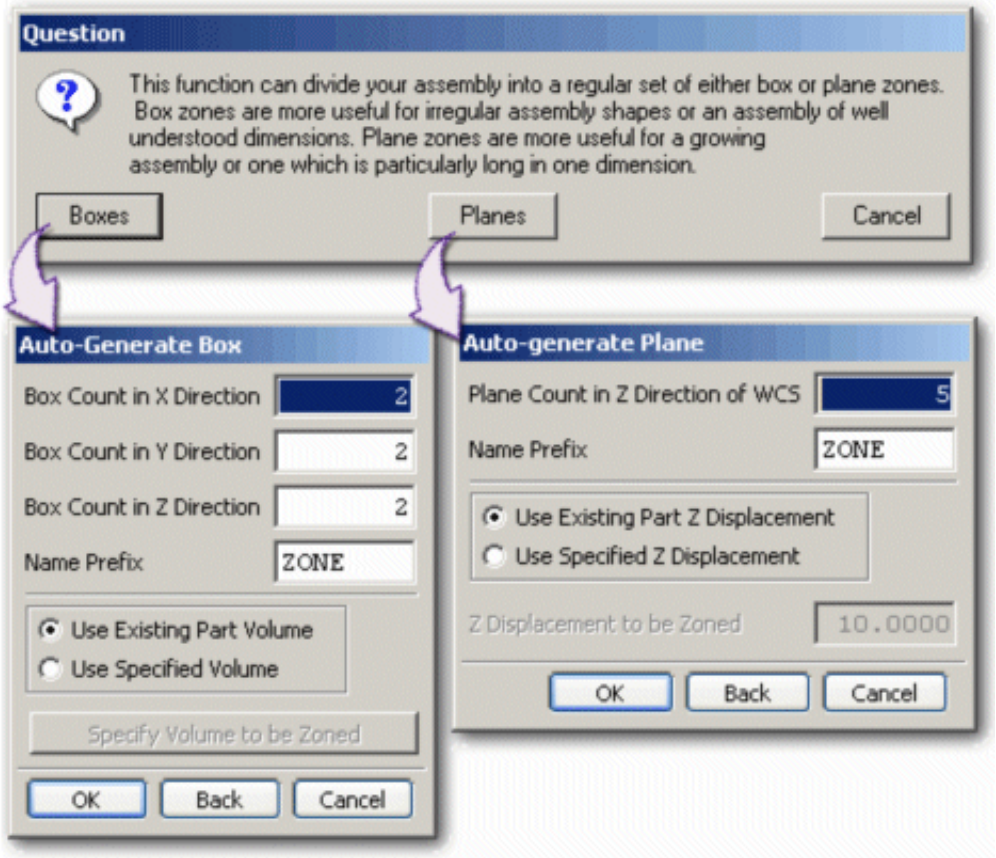
Delete Zone

Allows you to delete an existing zone. You should realize that by deleting a zone you will not be losing any geometry but just the zone definition.

2

Auto-Generated Zones

NX allows you to "Auto-Generate" zones to partition your assembly into regularly spaced box zones and/or plane zones. The system will generate the zones oriented relative to the WCS. When you choose this option, NX will prompt you with the following :



Auto-Generate Box:

Box Count - Allows you to input the number of box zones you want generated in the X, Y, & Z directions. (Relative to the WCS orientation)

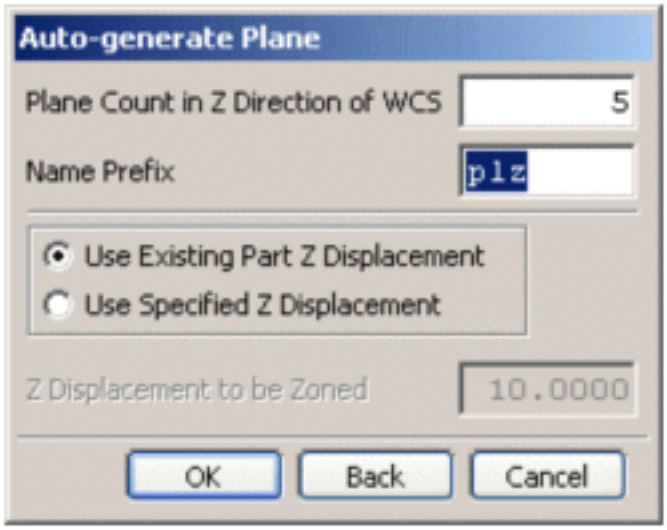
Name Prefix - Allows you to input a string of characters that will precede all the auto-generated zone names. For example, in the above dialog the system would generate eight zones named BZ1, BZ2, ..., BZ8.

Use Existing Part Volume - evaluates the minimum box envelope that would contain the assembly, then divides that volume up into the specified number of zones.

Use Specified Volume -when this options is toggled, you will then be required to define the size and location of a "total volume" box that will be subdivided. The orientation of the boxes will coincide with your current WCS.

2

Auto-Generated Plane



This option allows you to auto-generate equally spaced plane zones normal to the Z_C axis.

Plane Count - Allows you to input the number of plane zones you want generated in the Z_C direction.

Name Prefix - Allows you to input a string of characters that will precede all the auto-generated zone names. For example, in the above dialog, the system would generate five plane zones named PLZ1, PLZ2, ..., PLZ5.

Use Existing Part Z Displacement - This option evaluates the displacement of the assembly in Z_C direction, then divides that distance into equally spaced plane zones.

Use Specified Z displacement - This allows you to input the amount of Z_C displacement you would like to have divided up into plane zones. The system will prompt you to enter a distance along the WCS Z axis.

Activity: Component Filters with Zones

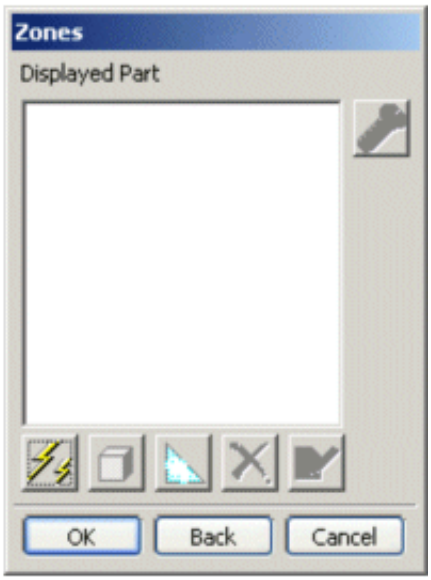
In this activity, you will be creating Zones and using them within Filters. You will become familiar with the creation and use of Zones and be able to utilize them within Filters.

Using this assembly, you will practice using zones to isolate portions of the assembly.

Step 1: Open part file assy2.prt from act2 directory.

Step 2: Create a Box Zone for the base component.

☐ Choose Assemblies→ Zones.



☐ Choose Create Box Zone

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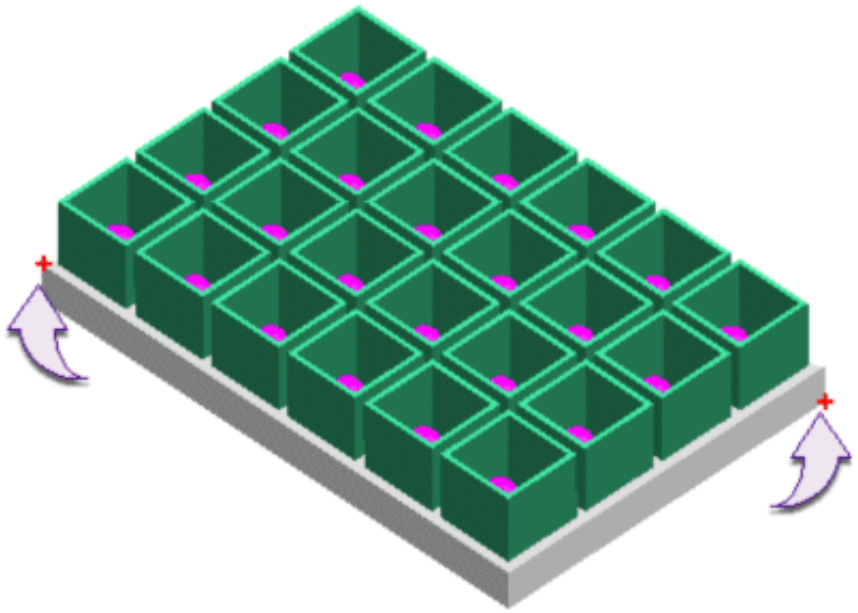
☐ Select the Two Diagonal Points option, and select the two diagonal corners of the base.



☐ Select the newly created zone and Rename it to "BASE".

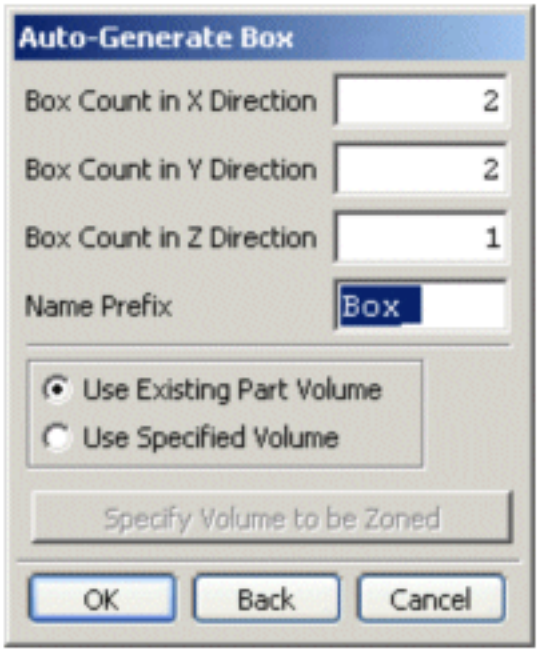
☐ Change your display to wireframe.

Step 3: Create some "automatic" box zones.

☐ Choose the Auto-generate Zones option and select Boxes from the next dialog.

☐ Enter the following values for your zone generation: Box Count in X Direction=2, Box Count in Y Direction=2, Box Count in Z Direction=1

☐ Make sure the Use Existing Part Volume option is selected, then enter "Box_" for the Name Prefix

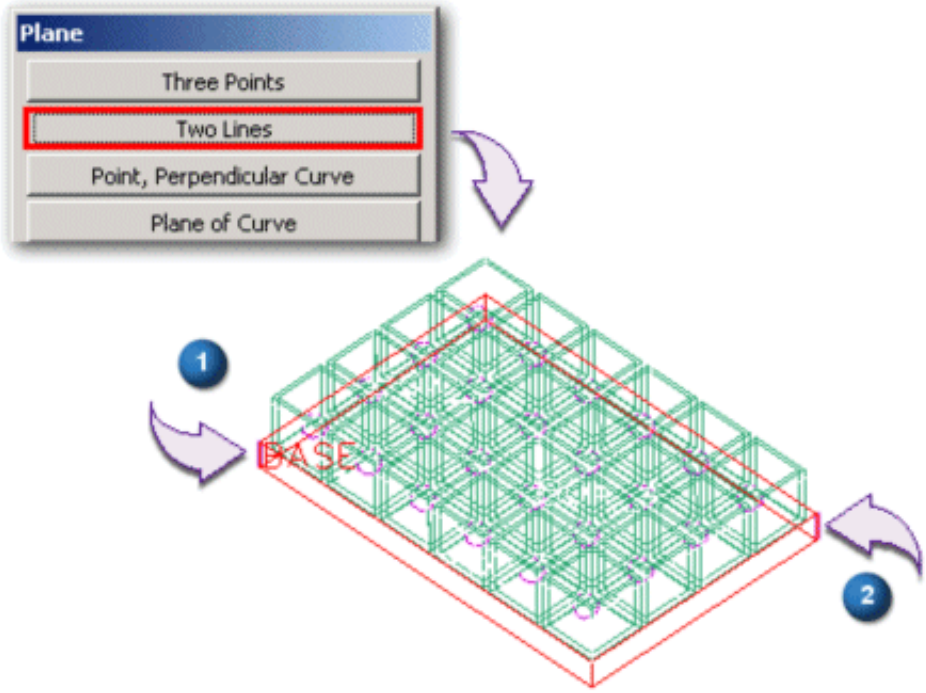


☐ Choose OK

Step 4: Create a plane zone.

☐ Choose the Define Zone option.

☐ Select the Two Lines option (1), and select the two diagonal opposite vertical edges of the base (2).



2

☐ Select the newly created zone and Rename it to "Cross Corners".

☐ Refresh your screen

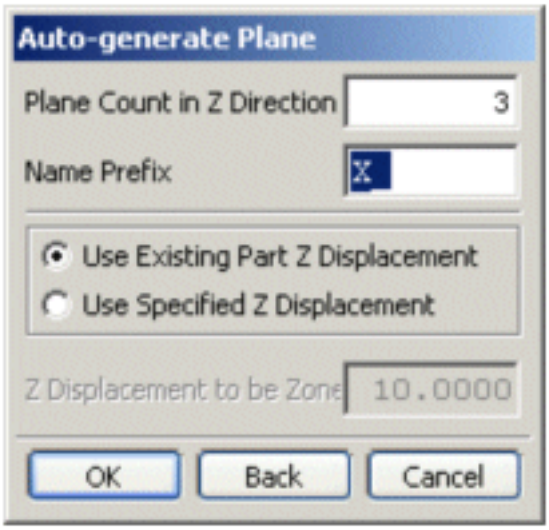
Step 5: Create some "automatic" plane zones.

☐ Choose the Auto-Generate Zones icon. Then select the Planes option from the next dialog.

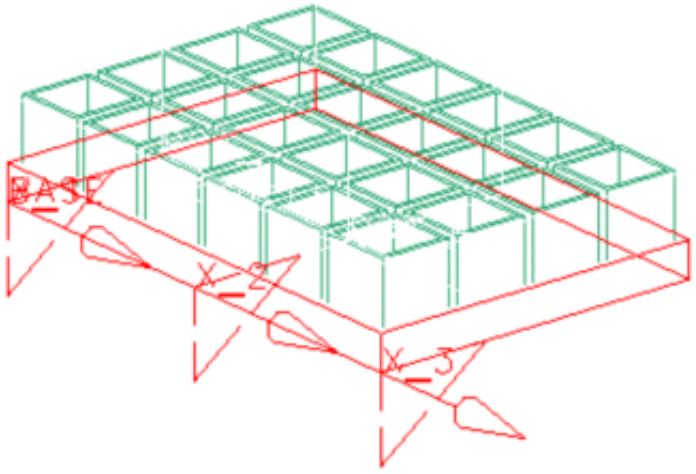
☐ Rotate the WCS so the new Z direction is along the longest, front edge of the base (absolute X direction).

☐ Enter 3 as the Plane Count in Z Direction of WCS,

☐ Enter the Name Prefix X_. Make sure to toggle ON Use Existing Part Z Displacement.



☐ Choose OK.



Step 6: Repeat the process, only create your plane zones along the absolute Y-axis.

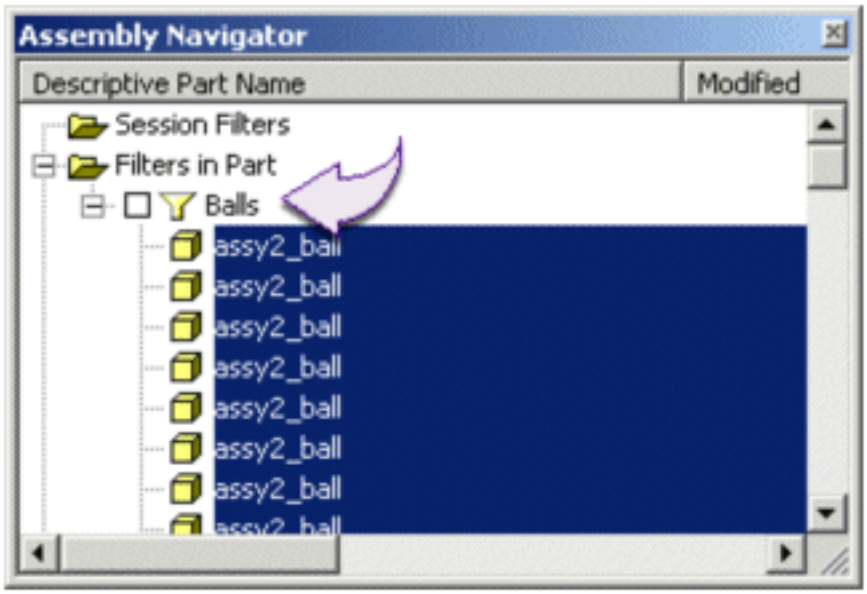
- ☐ Rotate the WCS so the new Z direction is along the shorter, rear edge of the base (absolute Y direction).
- ☐ Choose the Auto-Generate Zones icon. Then choose the Planes option from the next dialog.
- ☐ Enter a Plane Count of 3 again
- ☐ Enter the Name Prefix Y_ , then OK.

Step 7: Create Filters for the balls and boxes.

- ☐ Choose the Create Filter icon and enter then name " Balls".

2

☐ Select all of the assy2_ball components from the Assembly Navigator and drag them into the Balls filter. Be sure to get the ball at the bottom of the tree. (Use the Ctrl key.)



☐ Repeat the previous two steps except the filter should be named "Boxes" and should contain all the assy2_box components.

Step 8: Check your work, then save your assembly.

☐ Click on Check Box for each filter three times. First, to update the object count. Second, to blank the objects and third, to unblank the objects.

☐ Save your work, File→Save→Part.

These Filters and Zones are saved with the part. They can also be saved in a "bookmark" file so that they can be used with other assemblies if desired.

Bookmarks

Sometimes you may want to duplicate a current assembly state (e.g. which filters are applied, which components loaded) in a later NX session. A bookmark can help you do this without having to perform all the intermediate steps (such as reapplying the filters).

Bookmarks can record, communicate, and reuse the working context from a NX session. A bookmark file records:

- Currently defined assembly filters
- Current load options that relate to the configuration

The search directories and the load option for whether to use them (in UG/Manager, the revision rule will be recorded instead).

The Allow Substitution setting

Default reference sets

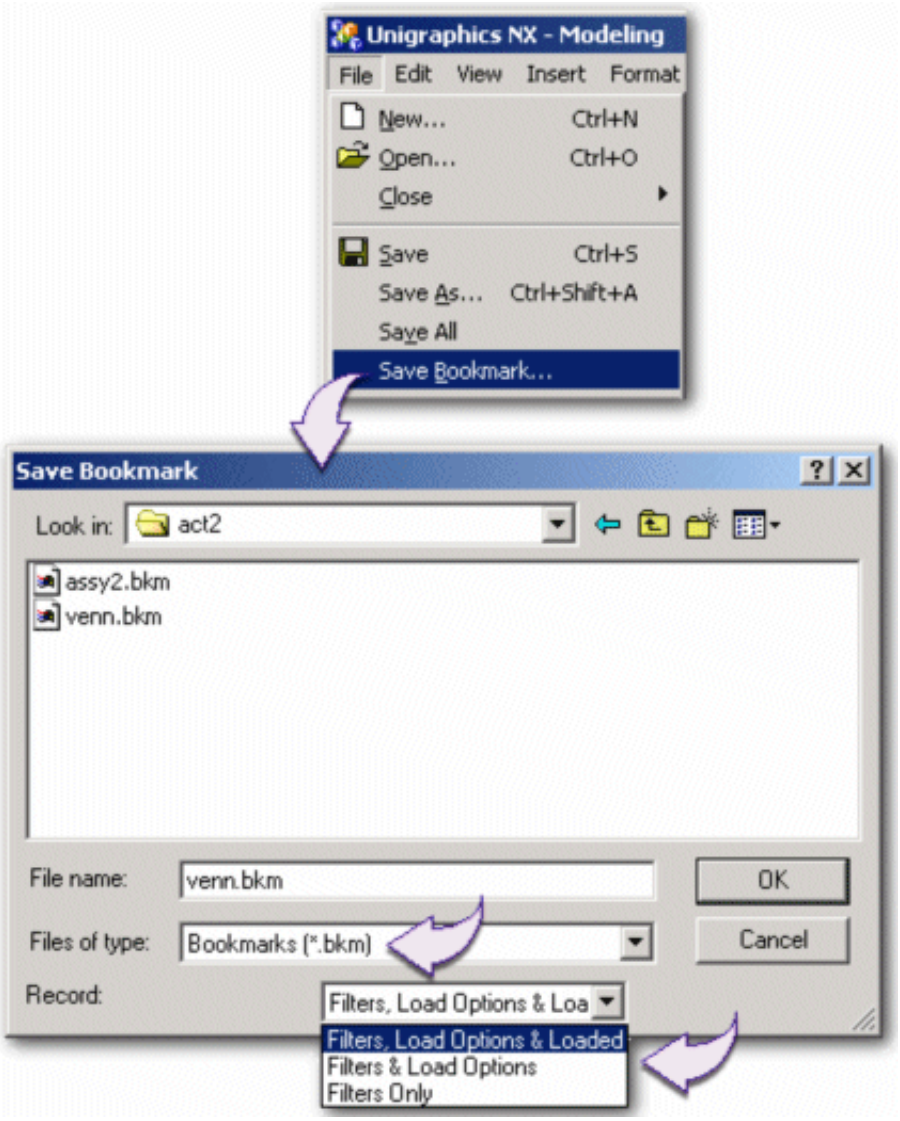
The setting of the Apply to All Levels reference set option

The current displayed part and the set of visible components

2

Creating a Bookmark

To save a bookmark, choose File→Save Bookmark. The Save Bookmark dialog appears, with the following options:



Within a Bookmark, you can either save Filters, Load Options and Loaded Components or only Filters and Load Options or only Filters.

Applying a Bookmark

To open and apply a bookmark, choose File→Open, choose Bookmarks (*.bkm) from the Files of type option menu, and select the bookmark. This will retrieve the working context of the session where you saved the bookmark. Depending on the information you recorded, the bookmark may load filters, reset load options, load parts, or change the displayed part.

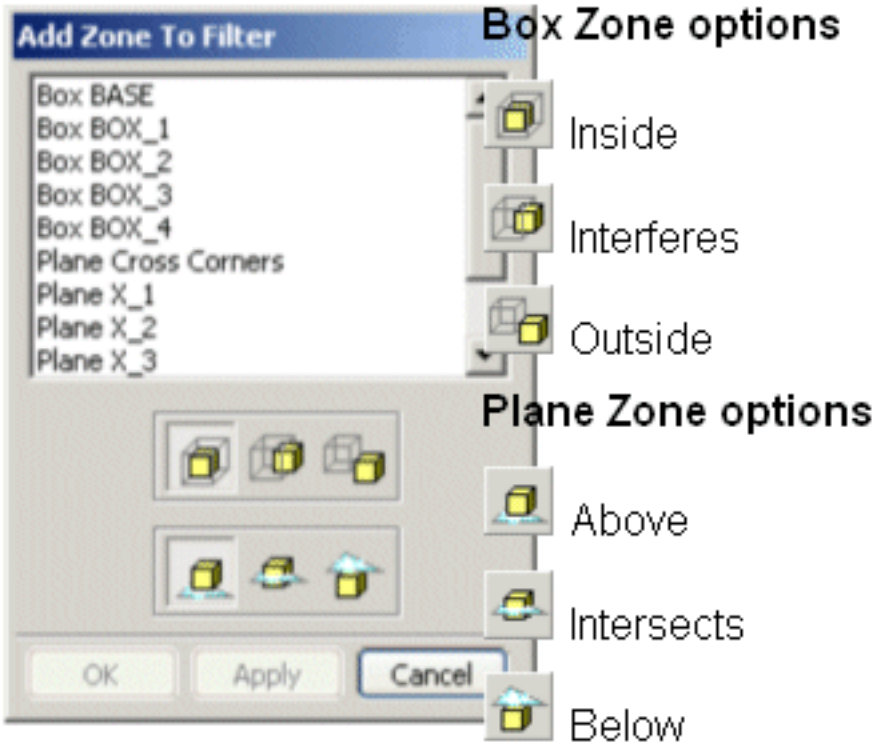
Save a Bookmark of Filters, Load Options and Loaded Components.

- Choose File→Save Bookmark.
- Enter your home directory: assy2.bkm
- Choose OK

Functional Filters - Add Zone to Filter

Zone filters select components according to their position relative to boxes or planes defined with the component filters zone options.

When you add a zone filter (e.g., by choosing the Add Zone to Filter icon from the Filtering toolbar), the Add Zone to Filter dialog appears. It has the following options:



Box Zone Options:

Interferes: Selects all the components that intersect or are inside the selected box zone. The bounding box of an object that touches the box zone is considered to intersect the box zone.

Outside: Selects all the components that are outside the selected box zone. The bounding box of an object that touches the box is not considered to be outside the box zone.

Inside: Selects all the components inside the selected box zone. The bounding box of an object that is inside and touches the box zone is considered to be inside the box zone.

In some situations, the behavior of a bounding box that touches a zone may vary.

Plane Zone Options:

Above: Selects all the components above the selected plane zone. The bounding box of an object that touches the plane is considered to be above the plane.

Intersects: Selects all the components that intersect the selected plane zone. The bounding box of an object that touches the plane is considered to intersect the plane.

Below: Selects all the components below the selected plane zone. The bounding box of an object that touches the plane is considered to be below the plane.

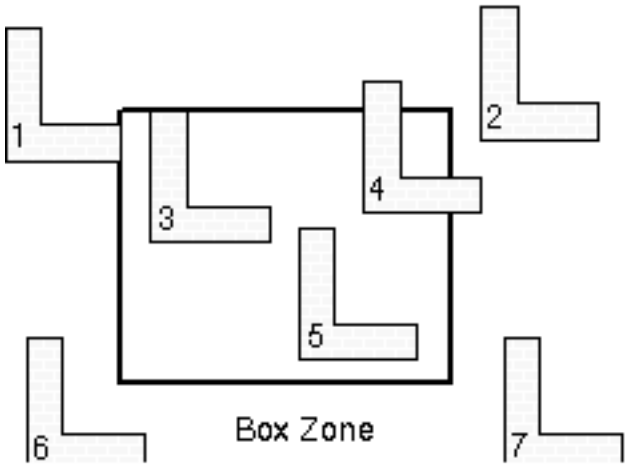
Add Proximity Filter

Selects all the components within a specified distance of the bounding box of a selected component. You can create a proximity filter by choosing the Add Proximity Filter icon on the Filtering toolbar.

You can modify the distance by using MB3 to select the within 10.0 in line of the filter and choose Edit from the popup menu.

Textbook Activities:

Inside - Selects all components that are entirely contained within the zone.



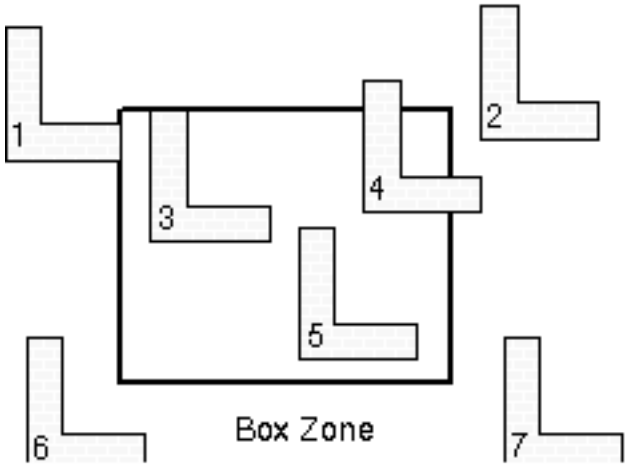
Circle Y or N if shape would be selected?

Box Zone

1) Y N
2) Y N
3) Y N
4) Y N
5) Y N
6) Y N
7) Y N

2

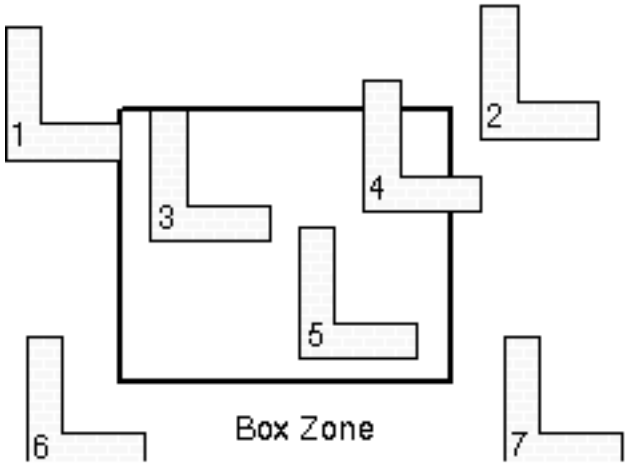
Interferes - This option will select all components whose bounding box interferes with the given zone.



Circle Y or N if shape would be selected?

Box Zone		
1)	Y	N
2)	Y	N
3)	Y	N
4)	Y	N
5)	Y	N
6)	Y	N
7)	Y	N

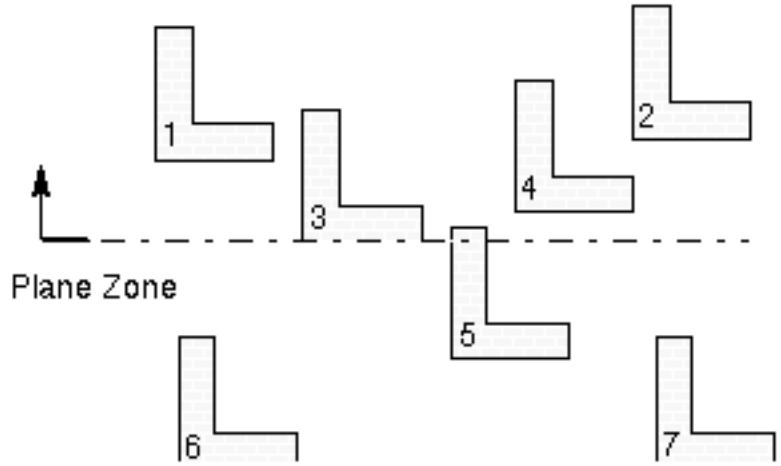
Outside - Selects all components that are entirely outside the zone.



Circle Y or N if shape would be selected?

Box Zone		
1)	Y	N
2)	Y	N
3)	Y	N
4)	Y	N
5)	Y	N
6)	Y	N
7)	Y	N

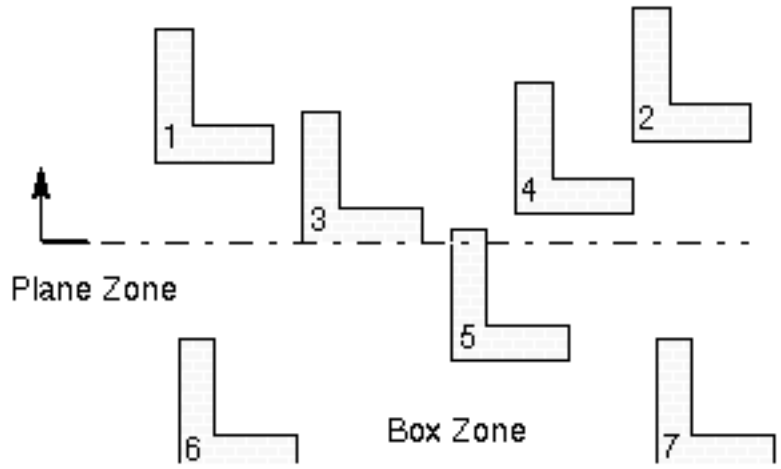
Above - Selects all components that lie completely above the specified plane zone. Above is relative to the vector normal direction of the plane. If the plane touches the bounding box, it will be considered above the plane.



Circle Y or N if shape would be selected?

Plane Zone		
1)	Y	N
2)	Y	N
3)	Y	N
4)	Y	N
5)	Y	N
6)	Y	N
7)	Y	N

Intersects - This option will select all components that intersect with the given plane zone.



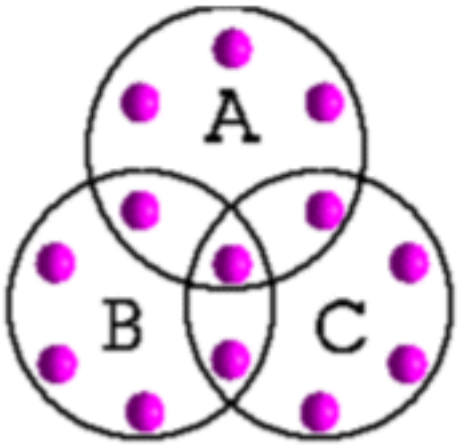
Circle Y or N if shape would be selected?

Plane Zone		
1)	Y	N
2)	Y	N
3)	Y	N
4)	Y	N
5)	Y	N
6)	Y	N
7)	Y	N

Below - Selects all components that lie completely below the specified plane zone. If the plane touches the bounding box, it will be considered below the plane.

The Work Part zone and the Proximity Zone are currently the only zones that will exhibit spatial associativity. In other words they will automatically update their size to reflect geometry changes when the components are edited and saved.

Organizing Filters Refer to the Venn diagram in the following examples and explanations.



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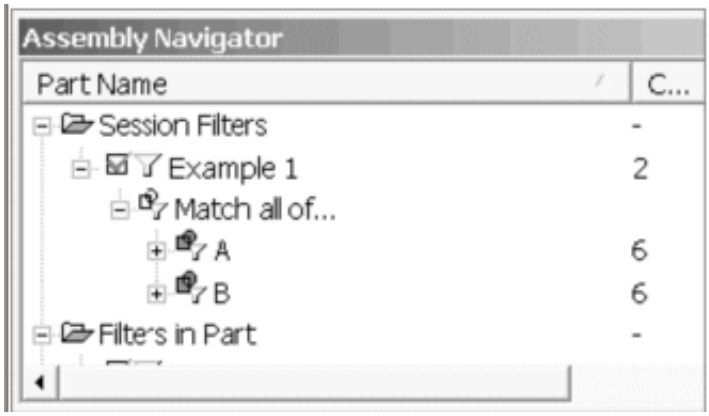
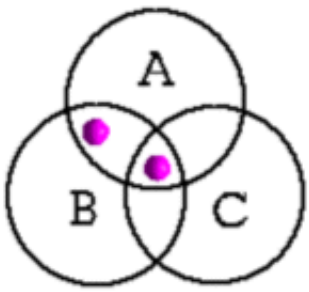
Match all of ... Filter

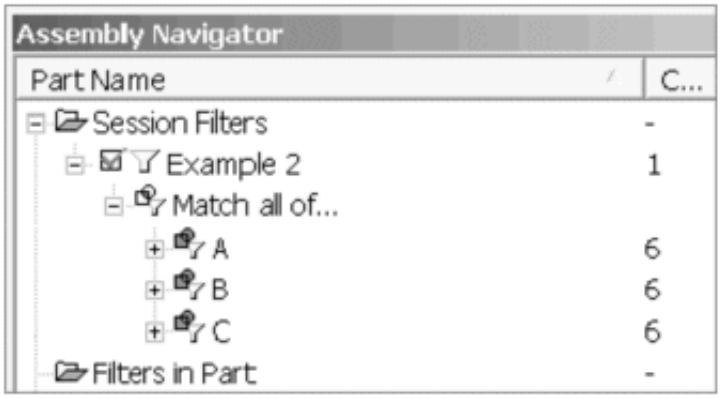
Selects components only if they meet all of the conditions specified under it and are not excluded by an Exclude filter.

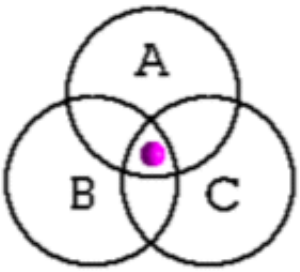
Example 1: Match all of A and B





Example 2: Match all of A, B and C

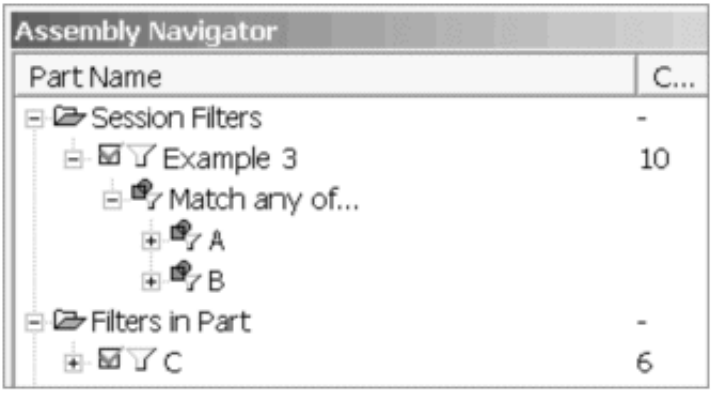


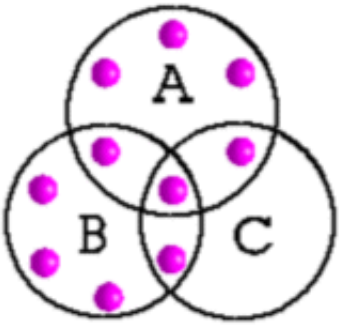


Match any of ... Filter

Selects components if they meet any of the conditions specified under it, as long as they are not excluded by an Exclude filter.

Example 3: Match any of A or B

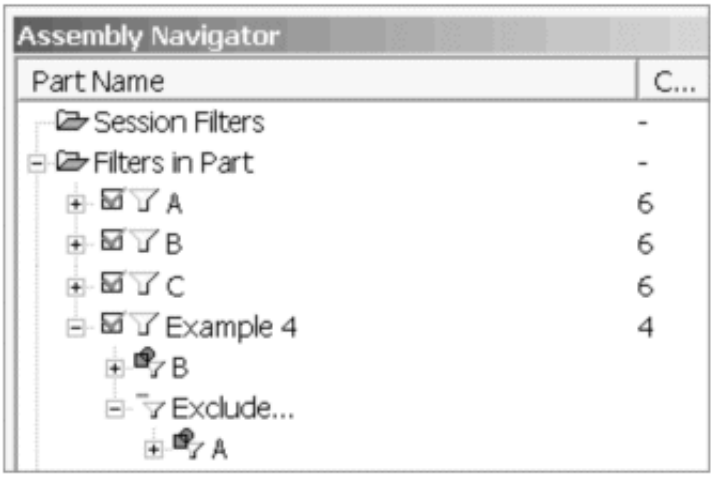


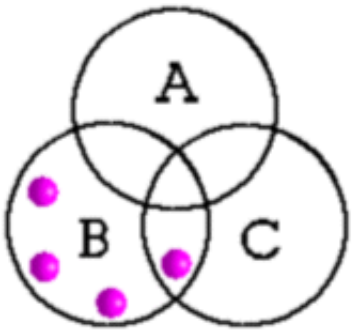


Exclude from Filter

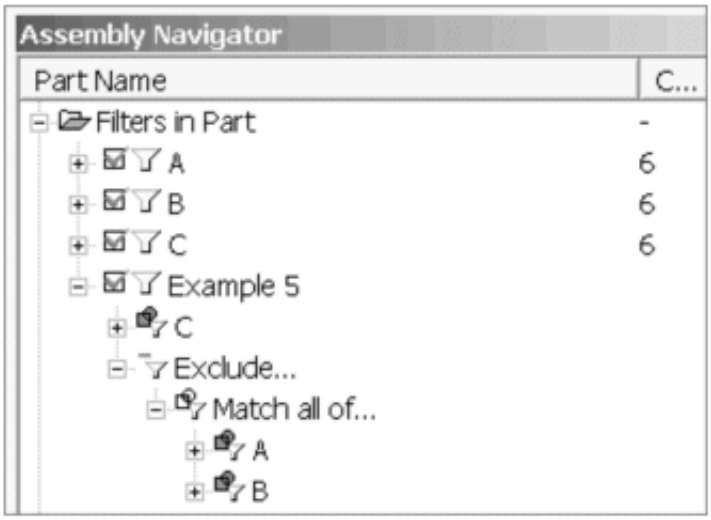
Lets you exclude selected components or portions of a filter.

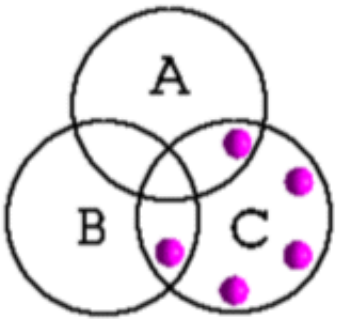
Example 4: All of B except A





Example 5: All of C except A and B





Tips and Techniques:

You can combine filters to create a more complex filter. Filters can use other filters as their input.

Filters are not selectable objects in NX; instead, they are used to select other objects.

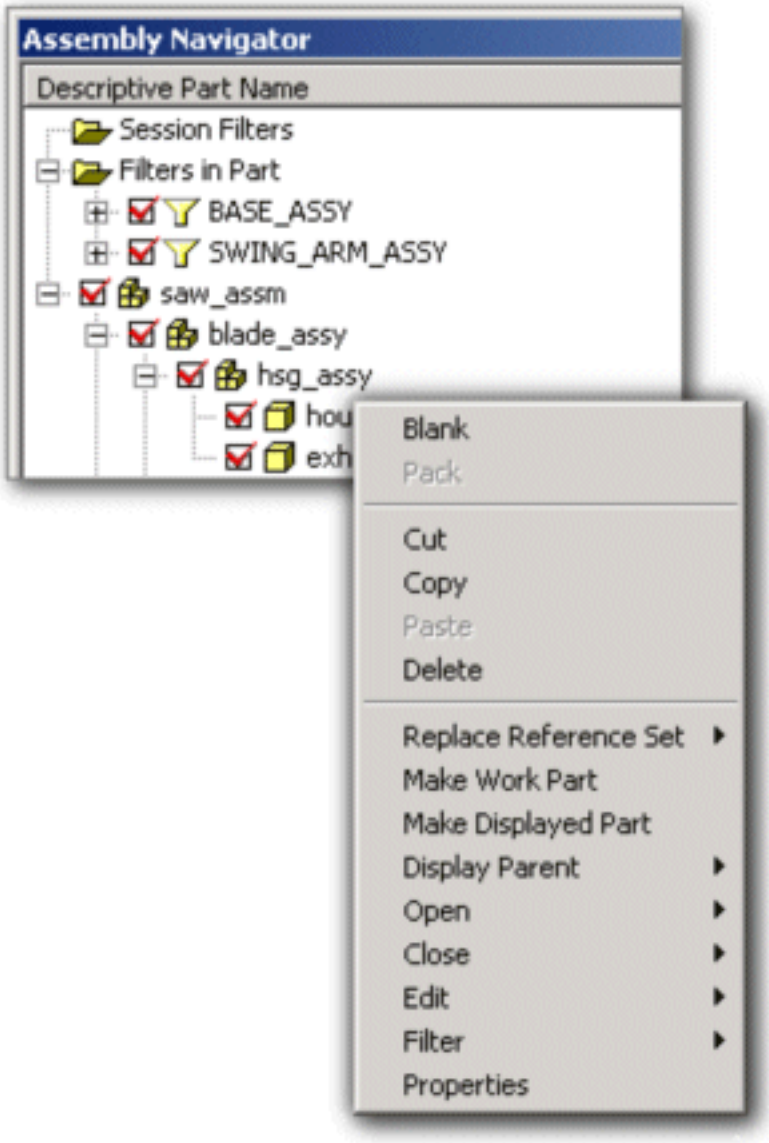
Double-clicking MB1 over a filter applies it. This updates the count and highlights the components included in the filter.

The checkboxes control the visibility of the components identified by the filter. If you check the box of a filter, all components in the filter are loaded

(if they are not already open) and made visible. Using the checkbox again will blank them.

Filter Popup Menus

When you place the cursor over a filter node in the Assembly Navigator and press MB3, a popup menu with the following options appears:



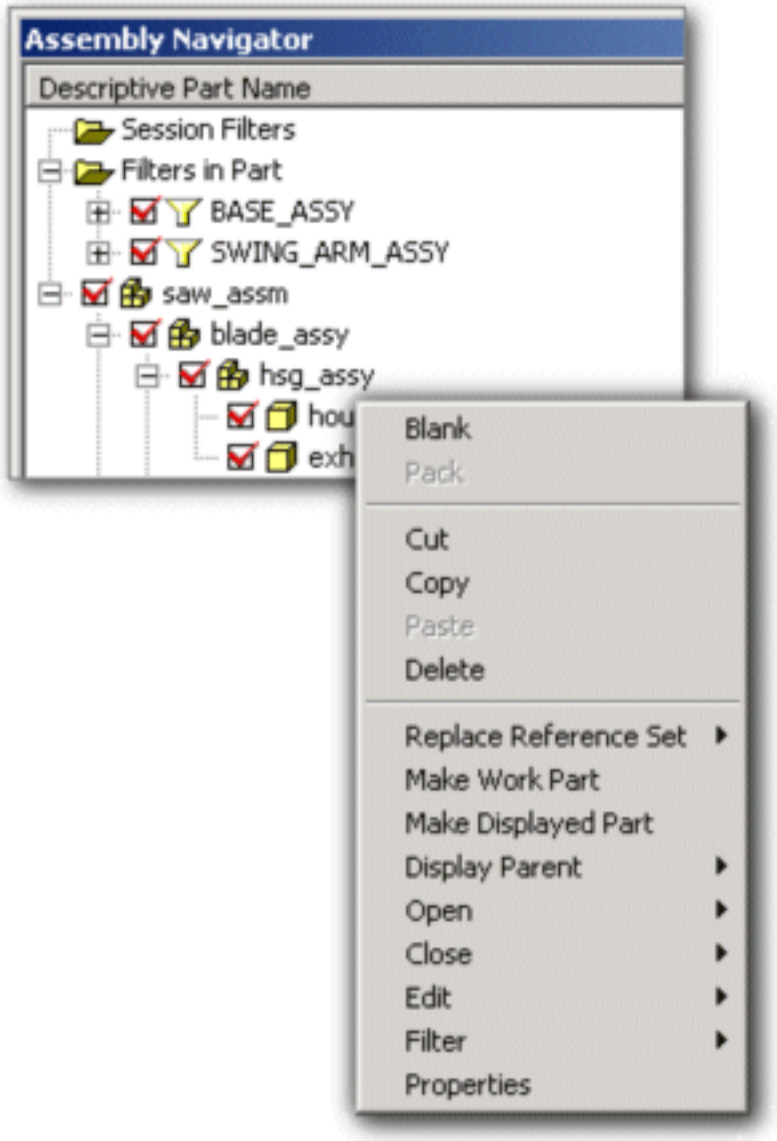
Additional options available under the pop-up menu for filters include:

Available when used with NX Manager:

- Check - In
- Check - Out

Component Option Popup Menus

When you place the cursor over a component node in the Assembly Navigator and press MB3, a popup menu with the following options appears:

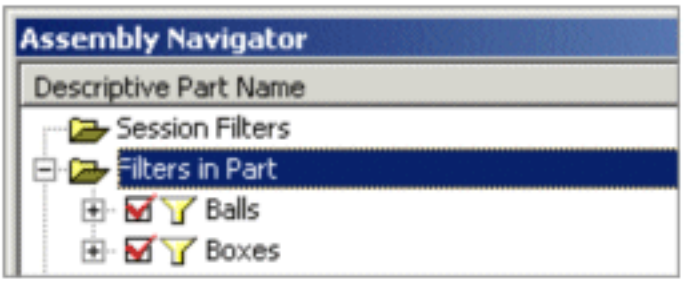


Activity: Component Filters with Zones, cont'd

The first filter you create will be designed to select only those parts that come in contact with the base.

Step 1: Continue using assy2.prt from the act2 directory.

☐ Select the top filter node.



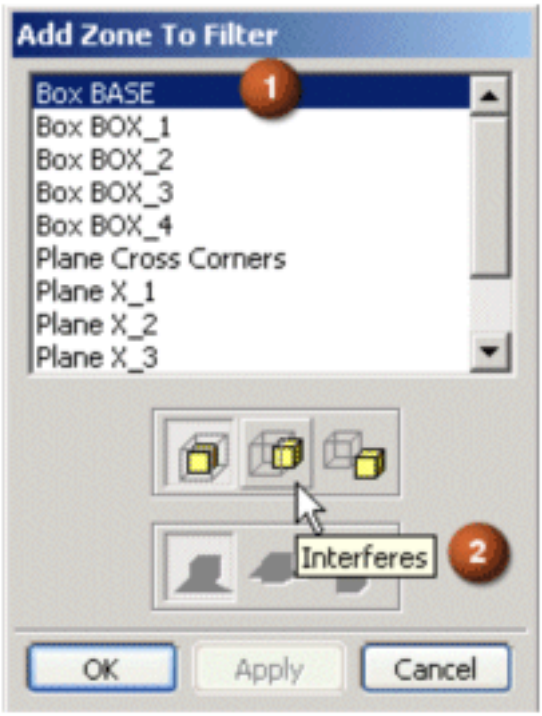
☐ Choose the Create Filter icon.

☐ Double-click MB1 on the New Filter text, type in "Touch Base" and press Enter.

☐ Choose the Add Zone to Filter icon.

☐ Select the Box BASE zone from the list (1).

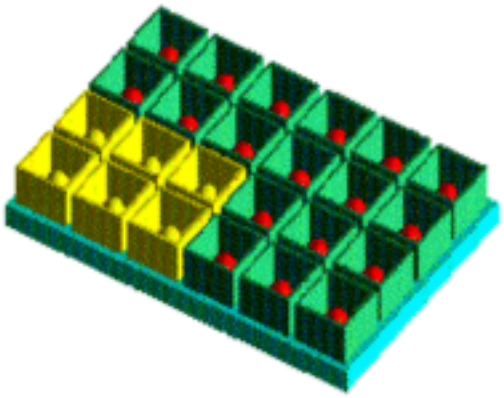
☐ Choose the Zone Comparison option of Interferes (2).



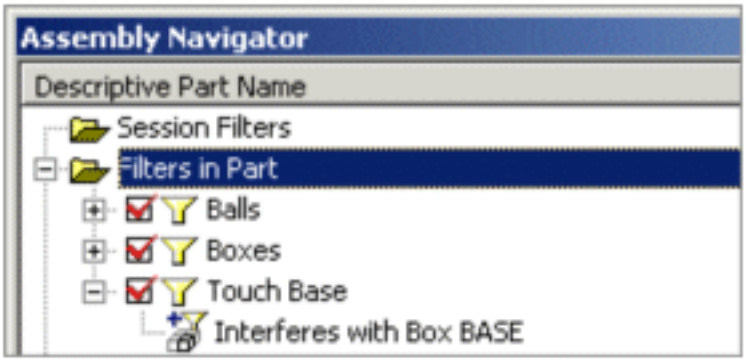
☐ Choose OK.

The zone creation technique used for the Base zone explicitly defined the zone size. If the base were to "grow" the zone would not do so automatically. A better method would have been to create a "proximity zone" for the base. That way any size changes made to the base would automatically be reflected in the base zone.

Step 2: Create a filter called Left Front that will select components in the left front quadrant of the assembly as shown below:



☐ Select the top filter node as the destination for the next filter.



☐ Choose the Add Zone to Filter icon from the Assembly Filtering Toolbar.

☐ Select Plane X_2.

☐ Select the zone comparison option of Below, then OK

☐ Change the name of the filter to Left Front.

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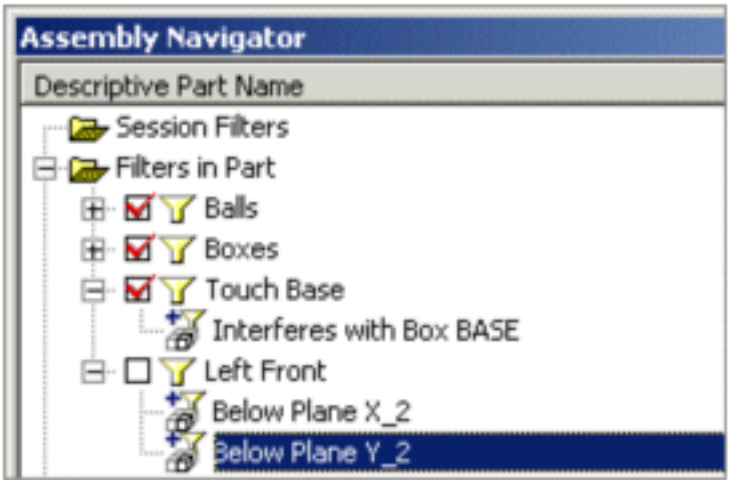
☐ With the new filter still highlighted, choose the Add Zone to Filter icon.

☐ Select Plane Y_2

☐ Select the Below comparator and choose OK



Without using a "Match..." condition in the filter, it will behave as if the two zones were combined in a Match any of...

Step 3: Click on the Left Front filter's checkbox 3 times.

Notice any components below either plane are selected. This is not the intent. You must add an organization level to this filter.

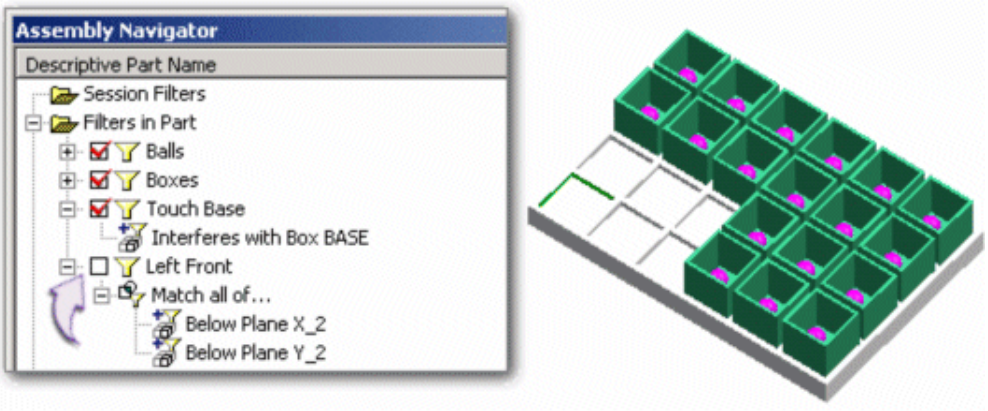
Step 4: Add a Match all of... clause.

☐ Select the two lines of the filter, Below plane X-2 and Below Plane Y-2 (Use the shift key)

☐ Choose the Match all of... icon from the toolbar.

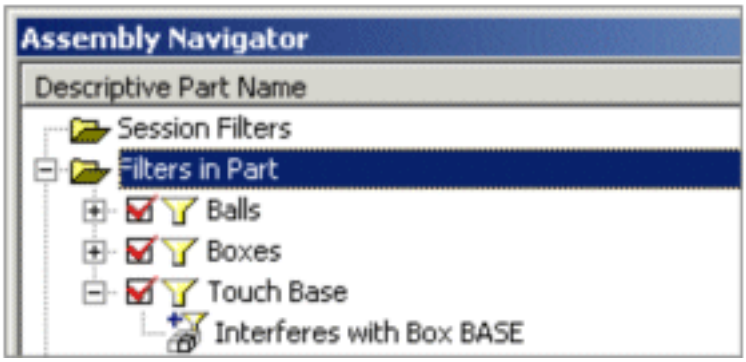
☐ Test the filter by double-clicking on the filter name

☐ Blank the contents of the filter using the checkbox.



Step 5: Create a filter called Right Rear Balls that selects only the balls in the right rear corner.

☐ Select the top filter node as the destination for the next filter.



☐ Choose the Add Zone to Filter icon.

☐ Choose Plane X-2 as the zone and choose Above, then choose OK

☐ Enter the name Right Rear Balls.

☐ With the Right Rear Balls filter still highlighted, choose the Add Zone to Filter icon

☐ Choose Plane Y-2 as the zone and choose Above, then choose OK

☐ Click MB3 on the Balls filter and select Copy from the pull down menu.

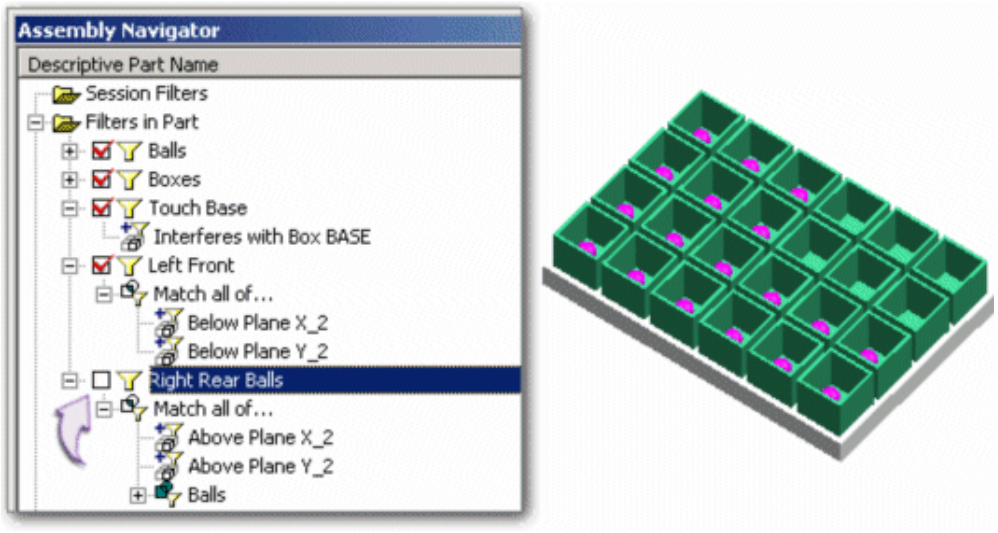
☐ Click MB3 on the Right Rear Balls filter and select Paste.

2

At this point if you activate the filter, you see the default behavior of Match any of... . Everything above X-2, above Y-2 and in the Balls set is selected. You need to put all three of these filters in a Match all of... level.

Step 6: Choose Above Plane X-2, Above Plane Y-2 and Match any of... .

☐ Select the Match all of... icon from the toolbar.



Take a few minutes and activate the filters you have created so far. Are you getting the results you expect? If not, ask your instructor to help explain your results.

Step 7: Save your assembly.

Using Filters With Attributes

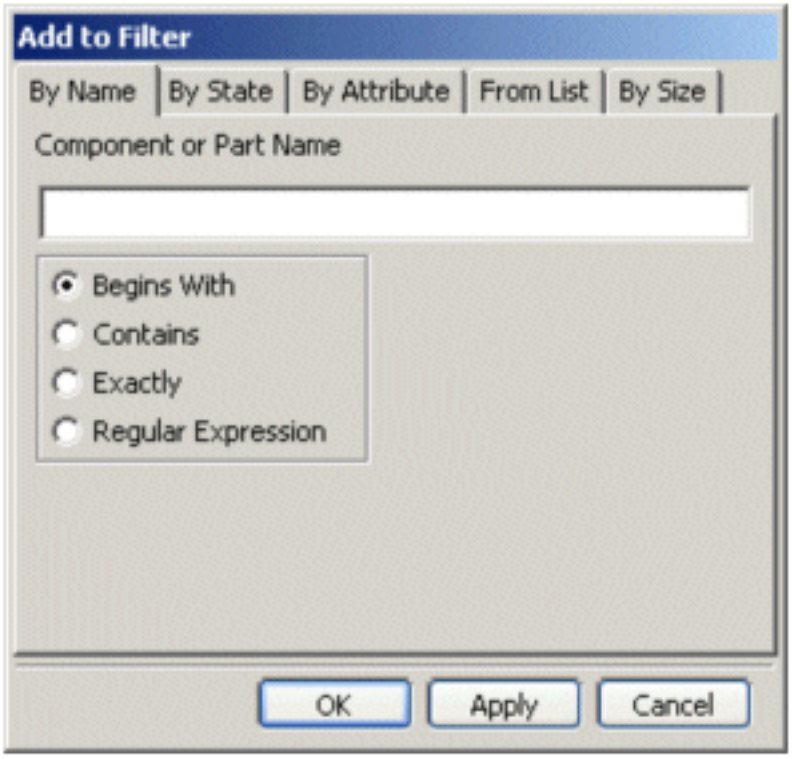
In order to use attributes within filters, they must be previously defined.

Attributes can be assigned to a reference set within a part, the component part and or the component object. If there are attributes assigned at all three levels, the system will read the object attribute first, the reference set attribute second and the part attribute third.

2

Add Attribute Search to Filter

You can create an attribute search filter with the Search Filter dialog, which appears when you choose the Add Attribute Search to Filter icon from the Filtering toolbar or by choosing MB3 and selecting Add to filter... .



The Attribute Filter will actively include components that meet the characteristics set forth by the filter. The term actively is used since any new components added to the assembly that match the defined criteria will be automatically included.

By Name

This option will allow you to input a string of characters and will include any parts whose name matches the character string.

- Begins With: includes all parts whose file name begins with the character string.
- Contains: includes all parts whose name contains the character string.
- Exactly: includes all parts whose name exactly matches the character string.

- Regular Expressions: uses a wild card system typically used by programmers. It is explained by the following:

  ? = 0 or 1 character

  * = 0 or more characters

  + = 1 or more characters

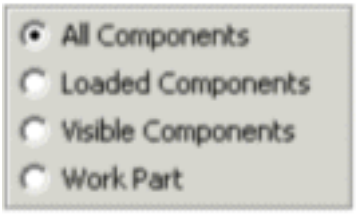
  . = any character

  | = or

  = set

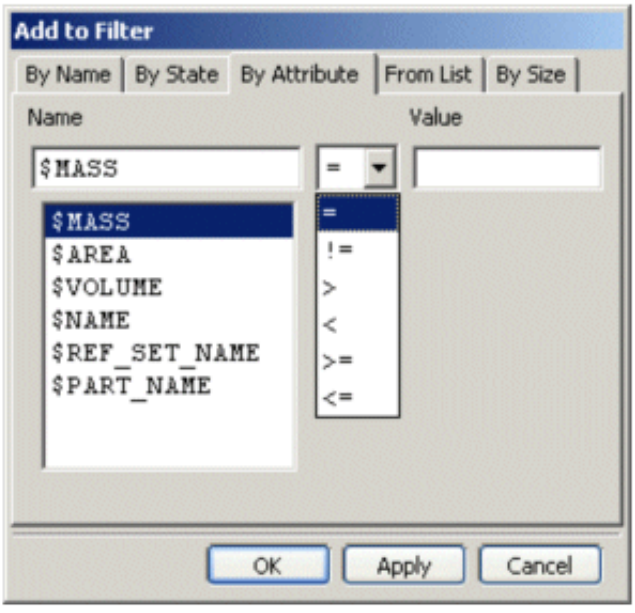
By State

This allows you to sort and include parts based on the descriptive status of:



All Components: all components of assembly included.
Loaded Components: loaded components included.
Visible Components: only visible components included.
Work Part: only the work part is included.

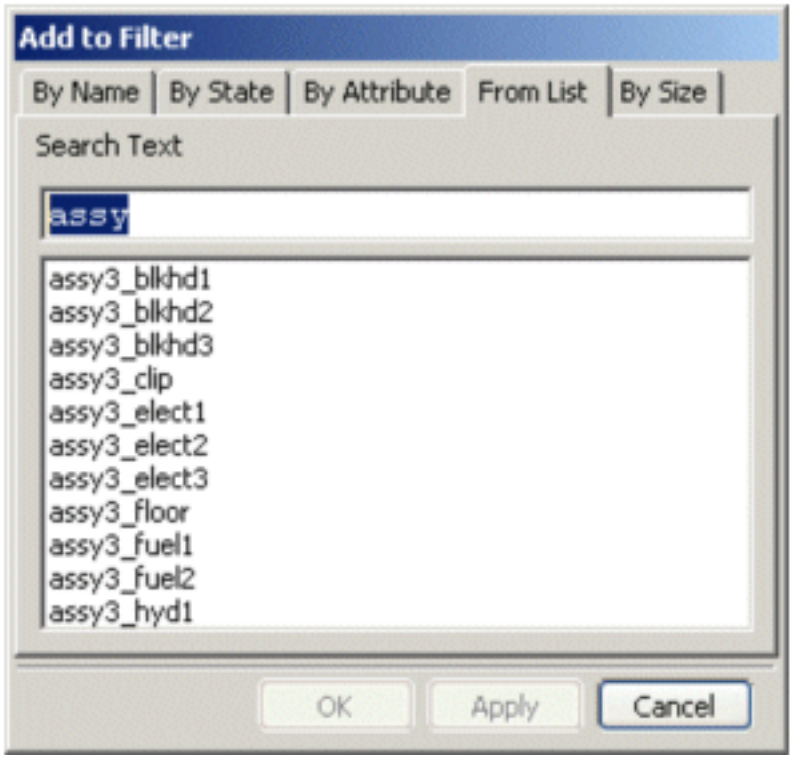
By Attribute



This option allows you to select an Attribute from the list, followed by an operator and a value. All components that meet that criteria will be included by that filter. For example; you could find all the components whose calculated mass is greater that .05 pounds.

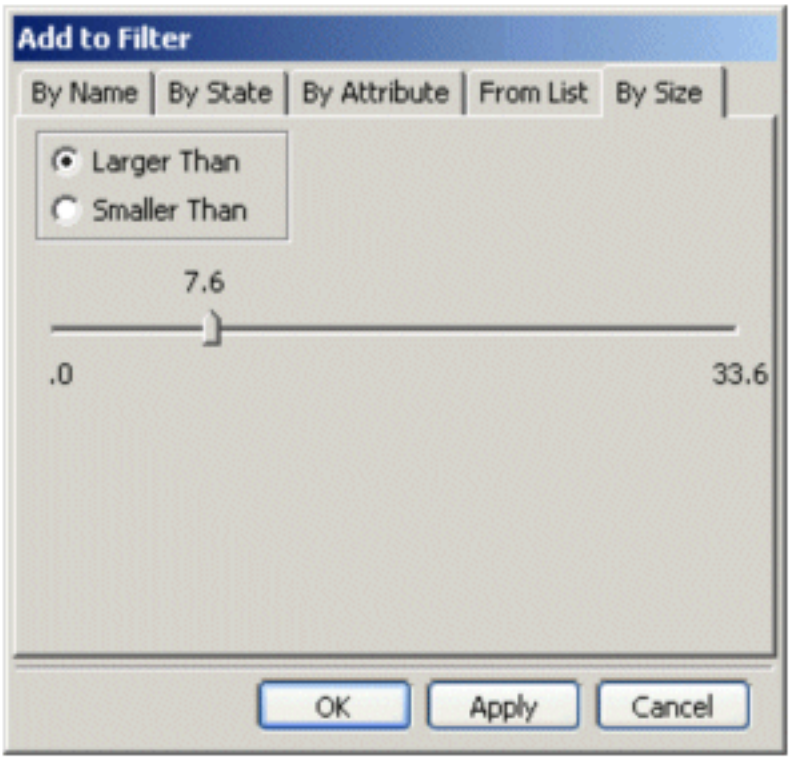
Only system defined attributes will appear in the list. User defined attributes will need to be manually entered like: MAT = aluminum.

From List



This option allows you to select from all of your components which to be included. The Search Text field allows you to filter the list based on your inputted text string.

By Size



This option allows you to include components based on their size which is measured as the largest diagonal of the bounding box for each component. You can toggle between Larger Than and Smaller Than and adjust the value using the slider bar.

Update Structure

You might be asking yourself the question: How does my assembly know what attributes an unloaded component has or what zone it resides in or how large is its bounding box? How is NX able to filter components based on information found in unloaded components?

The answer is that an assembly file stores a "snapshot" of the structure beneath it. For each component in the assembly, the snapshot stores the attribute and bounding box information. This way it is possible to open parts based on their attributes.

However, it is important to remember how this snapshot of the assembly structure, bounding box, and attributes is updated. The update can occur in two ways:

- If you open the components in that assembly, the assembly updates its snapshot of that part in the assembly structure as well as any changes to the bounding box or attributes.

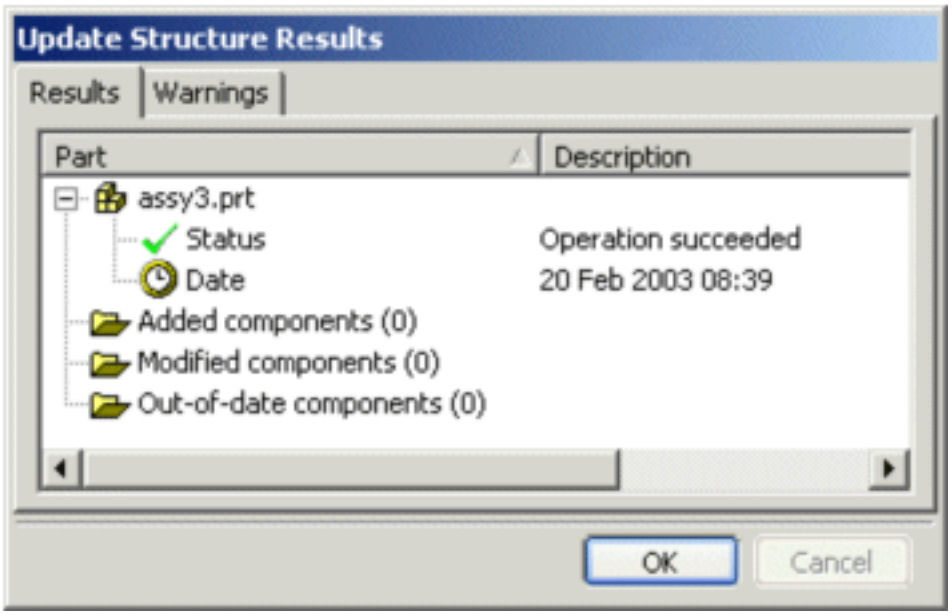
- When you choose Update Structure from the Tools→ Assembly Navigator→ Update Structure pulldown menu or choose the icon , the

assembly will update its snapshot by querying any unloaded components in the assembly which have changed since the assembly was last saved.

You can see that this updating procedure is important if you want to reliably execute a filter using zones or attributes. In fact, it is highly recommended that you run Update Structure before using a zone or attribute filter, unless all of the assembly is loaded.

Another benefit to Update Structure is to bring your Assembly Navigator up to date. For instance, you load an assembly with load components set to No Components. The Assembly Navigator will show you all components that made up that assembly the last time it was saved. However, a sub-assembly below it may have had components added or removed. The top assembly would not know that until the sub-assembly in question was loaded or until you chose to Update Structure.

Upon executing the Update Structure command, the system will generate a report detailing how many components were added, modified, or out-of-date.



Activity: Component Filters with Zones, cont'd

Step 1: Create a filter that selects parts that are in 2 inch proximity to the work part.

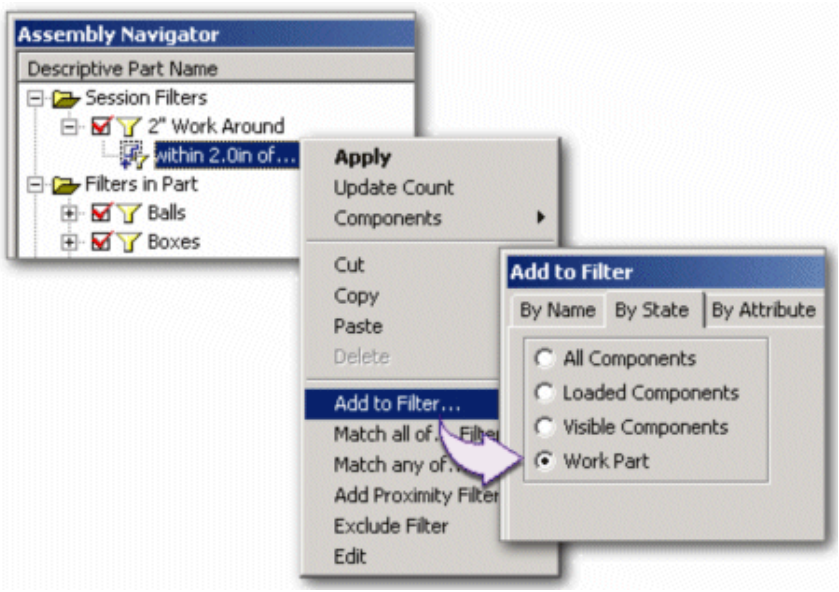
☐ Highlight the folder Session Filters to designate the destination of the new filter.

☐ Choose Add Proximity Filter.

☐ Change the name of the filter to 2" Around Work.

☐ Highlight the within 10.0 of... and use MB3→Edit to change the value from 10 to 2.

☐ Highlight the within 2.0 in of ... use MB3 → Add to Filter..→ By State →Work Part → OK.



Experiment with this filter. Change your work part to one of the components and click on the check box to blank and unblank the "filtered" parts.

A common mistake made when working with a filter containing Work_Part is to try and use or preview it when the work part is set to the displayed part. When this is done, nothing happens.

Next, we will make use of these filters.

Step 2: Create a filter named Exclude 2" Around Work that will leave only those parts near the work part displayed after its execution.

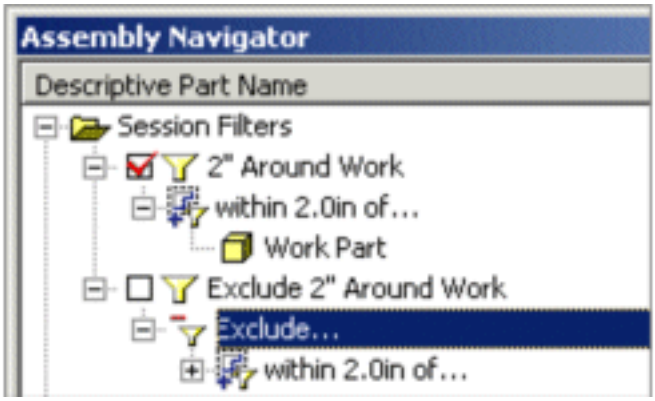
2

☐ Click MB3 on the 2" Around Work filter and choose Copy

☐ Choose the Session Filters node in the Assembly Navigator as the destination, click on MB3 and choose Paste

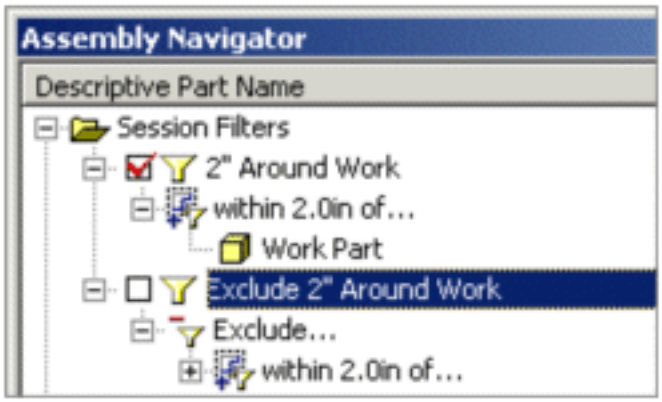
☐ Change the name of the copied filter to Exclude 2" Around Work and expand the filter.

☐ Exclude the components. Select the within 2.0in of the new filter and then choose the Exclude from Filter icon from the tool bar.



Now you need to add a set of components from which the "2 in around work" will be excluded.

☐ Select the filter "Exclude 2" around work, MB3→ Add to Filter → By State→ All Components → OK.



☐ Change the Work Part to any one box or ball.

☐ Blank and Unblank the 2" Around Work filter to see its results.

☐ Save Bookmark as before (you can overwrite a previous Bookmark).

☐ Continue with your part in the next activity.

Challenge: See if you can create, edit or copy existing filters to blank everything except balls within 3 inches of the work part and below the cross_corners plane zone.

Activity: Create a Bookmark

The intent of this activity is for you to practice the bookmark functionality.

Step 1: Save your assembly and, if you have saved your bookmark, close all parts.

When you saved your bookmark, the record method was set to "filters, Load Options and Loaded Components". The bookmark remembers the loaded parts so rather than open the assembly part, you only need to open the saved bookmark.

Step 2: Choose File→Open

Step 3: In the open part dialog, change Files of type to Bookmarks (*.bkm) and select your saved bookmark.

Notice the assembly opened and all of your filters are again available. If your assembly was already open and you opened this bookmark, the system would load the parts (if necessary) and retrieve your filters.

Step 4: Close all parts.

Step 5: Reset the Files of type option and open the assy2 assembly, without loading its components.

- ☐ Choose File→Open
- ☐ Set the Files of type widget back to NX Part Files (*prt).
- ☐ Click the Do Not Load Components checkbox and open the assy2 assembly

Notice all of the components show in the Assembly Navigator, but you do not get the filters.

Step 6: Using File→Open, open the bookmark again

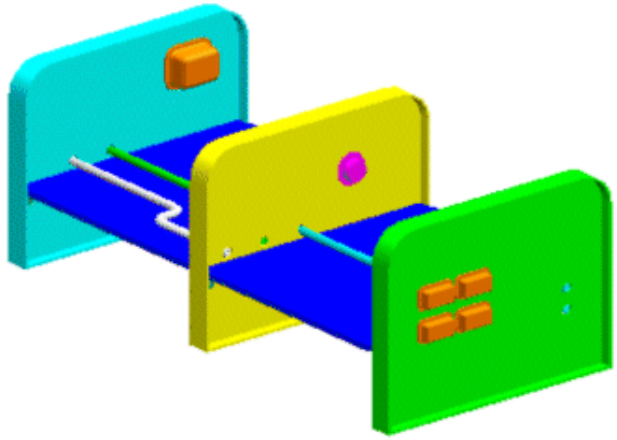
The Bookmarks now appear and the components loaded when you saved the bookmark (all of them) are opened.

If the bookmark had been saved with the record option set to filters only, the components would not have been loaded in Step 6.

Step 7: Close your part file.

Activity: Perfect Practice Makes Perfect Skills

In the following activity, you will practice using what you have learned so far. You will be creating Zones and Filters.



You will be given a list of tasks to devise solutions for. The specific steps to accomplish those tasks will be given at the end of the activity. You should attempt to work through these tasks on your own to maximize the learning process. As you finish each of the tasks, compare your solution with the one provided.

The assembly you will work with has some user defined attributes already assigned.

Each component has been classified as to whether or not it can be replaced in the field:

Step 1: line_maint =1 means field replaceable :

Step 2: line_maint =0 means not field replaceable
Each component has also been classified as to its function as follows:

usage=structure
usage=hydraulic
usage=fuel
usage=electrical

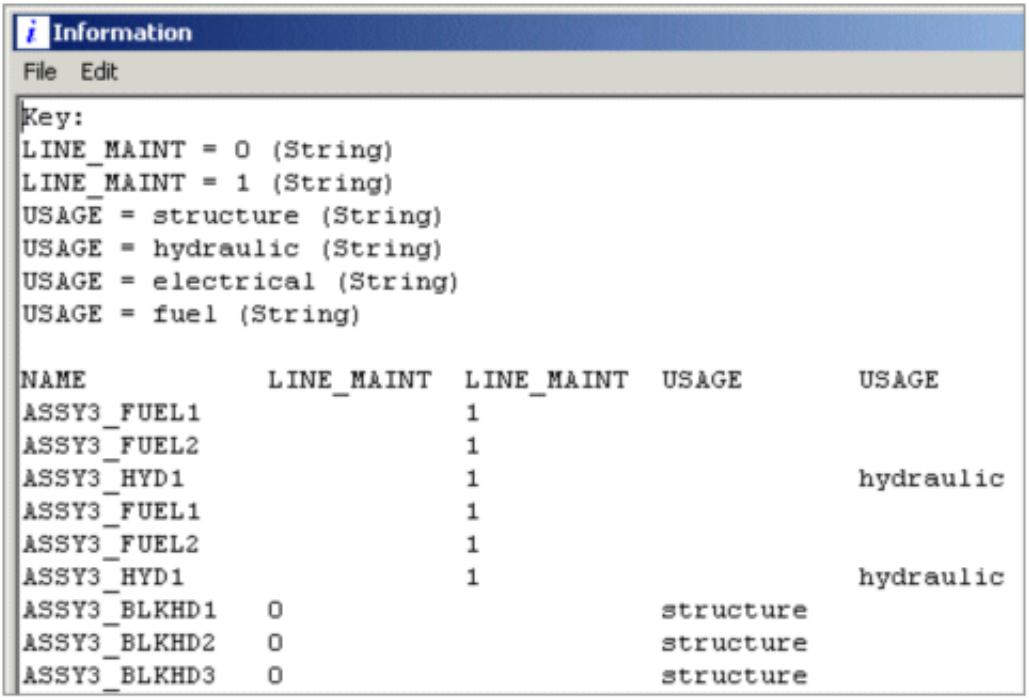
Step 3: Open assy3.prt from the act3 directory. If desired, list the components, list the attributes, and shade the part in order to become familiar with the design.

Step 4: **Optional:** Create an Attribute Report listing the existing user-defined attributes assigned to the components of the assembly.

☐ Choose Information→Other Attributes Report.

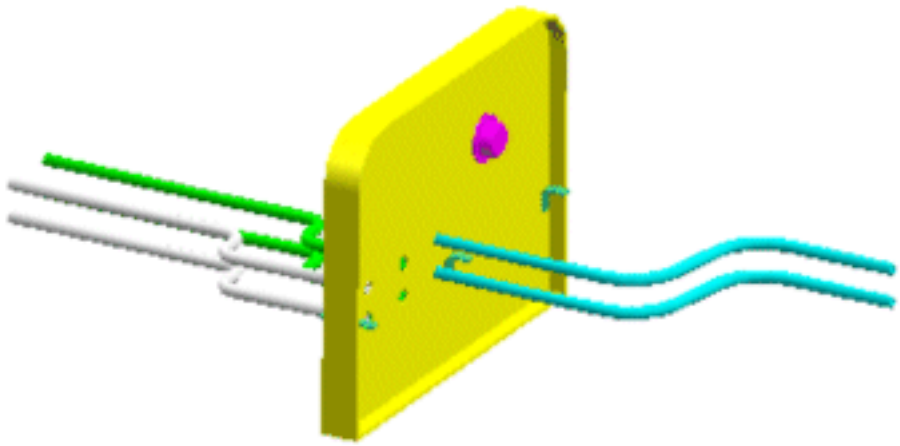
2

- ☐ Next, select Multiple Key Report, from the Report Writer dialog.
- ☐ Choose Type then highlight Component only, choose OK. Choose Select All.
- ☐ Choose the Variable Length Character option.
- ☐ Enter USAGE for the Key Title, then OK.
- ☐ Enter structure for the Sting Value, then OK.
- ☐ Repeat using the Key Title of USAGE and assign the String Values of hydraulic, fuel and electrical (3 more entries).
- ☐ Next, enter LINE_MAINT as the Key Title, then OK
- ☐ Enter 0 as the String Value, then OK
- ☐ Repeat using the Key Title of LINE_MAINT and assign another string value of 1
- ☐ OK when finished designating the multiple key entries, then OK three time to invoke the report.



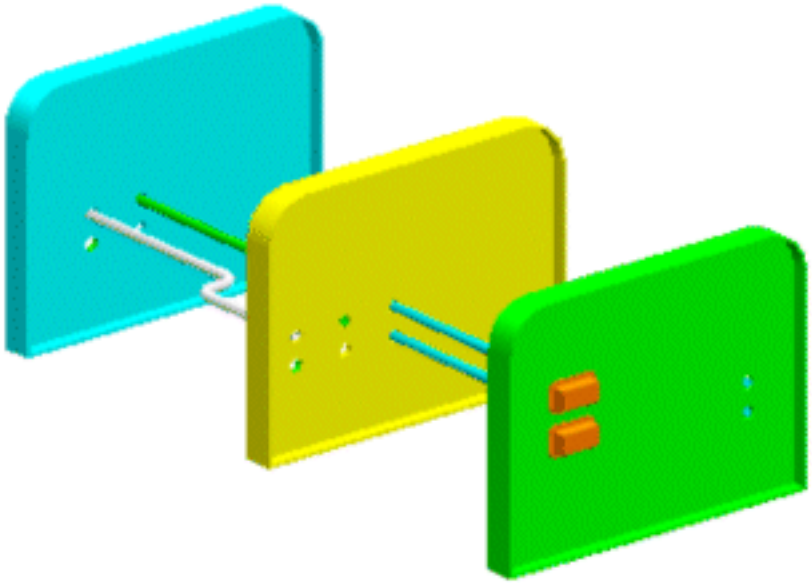
Task 1

Using whatever means necessary, devise a way to open only those components which are attached to assy3_blkhd2.prt. This does not include the floors.



Task 2

This assembly has redundancies built in. Create a filter that will display only components that are part of the primary system. All components that are members of the primary system can be seen below:



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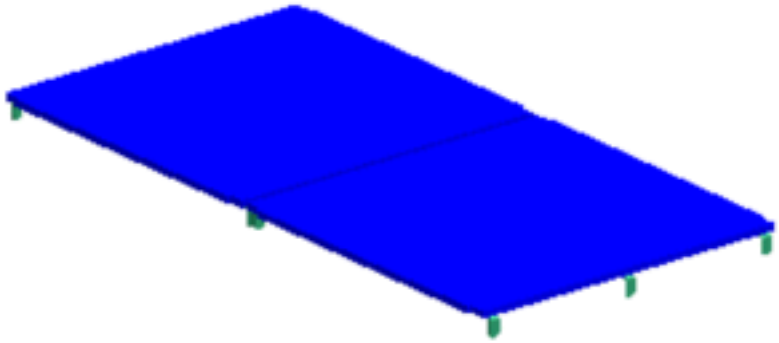
Task 3

Create a process using component filters etc., that will load only field replaceable structural components.



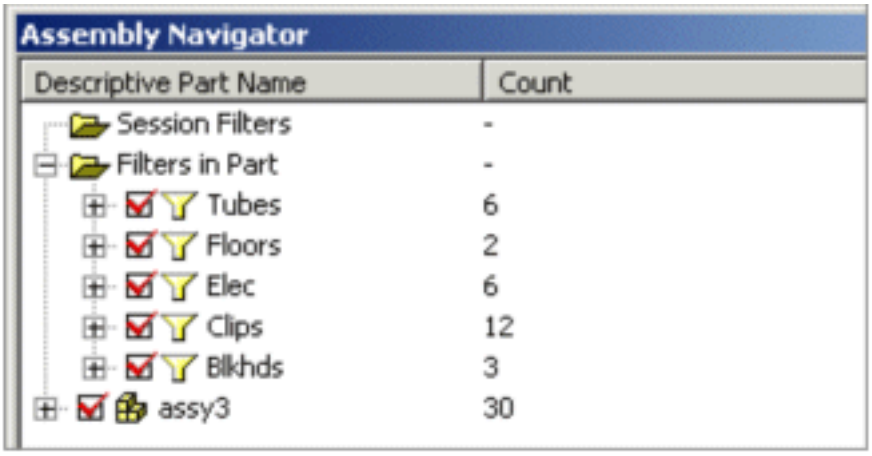
Solutions For Above Tasks

Each of the assigned tasks has several solutions. The solutions you found may be better or easier. If so, congratulations - pat yourself on the back!

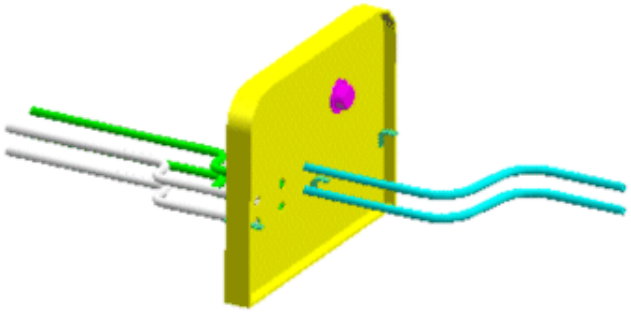
It is a good idea to group components under several filters and use them in the solutions to the previous tasks.

Group your components

Create the following filters:
Tubes: containing all fuel and hydraulic tubes.
Floors: containing the floors
Elec: containing all electrical boxes
Clips: containing all the clips
Blkhds: containing all bulkheads



Task 1 Solution: Assuming the assembly is open and all components are visible:



Step 1: Create a Filter that isolates the desired components.

- ☐ Highlight Session Filters and choose Add Proximity Filter
- ☐ Double-click the name of the filter (New Filter) and enter the name task1 (or MB3 on New Filter and choose Edit).
- ☐ Choose Edit

2

☐ Replace the distance 10.0 with 0 (zero) and press Enter.

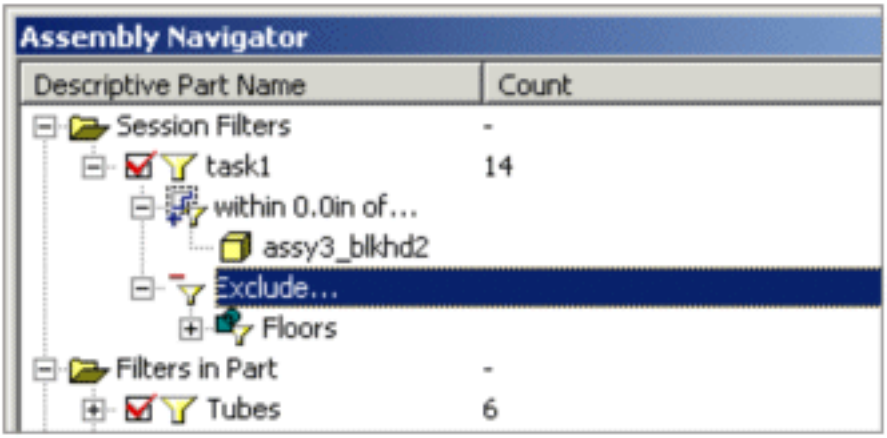
☐ Click on assy3_blkhd2 and drag it into the new Filter on top of within 0.0 in of....

Step 2: Exclude the floors from the new filter

☐ Click MB3 on the Floors filter and choose Copy.

☐ Click MB3 on the task1 filter and choose Paste

☐ Select the Floors line of the filter and choose the Exclude from Filter icon on the toolbar.



Now test the filter using the following steps:

Step 3: Close all components.

☐ Select the assy3 assembly node, MB3→ Select Assembly → MB3 → Close Assembly

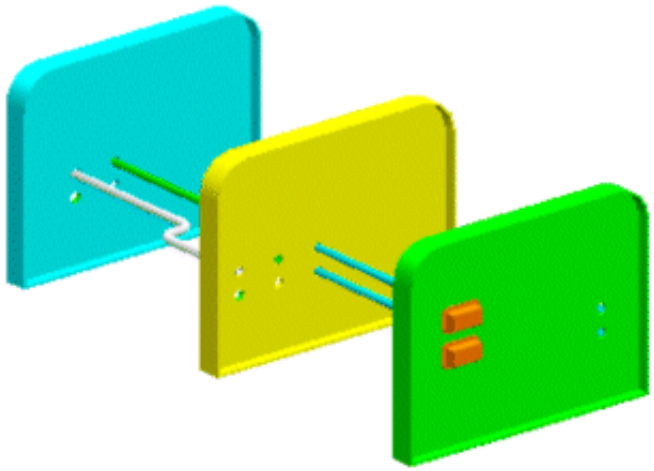
Step 4: Open the components defined by the task1 filter.

☐ Click on the task1 checkbox

Step 5: Save a Bookmark named assy3.

☐ Choose File→Save Bookmark→Record = Filters Only→ File Name = assy3.

Task 2 Solution:
Again, assuming the assembly is open and all components are visible:



Step 1: Create a filter with the primary system components.

- ☐ Select the Filters in Session node and choose the icon for Create Filter from the tool bar.
- ☐ Rename the filter to task2

An easy way to do this is to set your global selection filter to component, select the desired components from the graphics screen, and then choose the icon for Find in Navigator. These options are found in the Selection tool bar.

Step 2: Select and drag the desired components into the filter.

Step 3: Isolate the filtered components.

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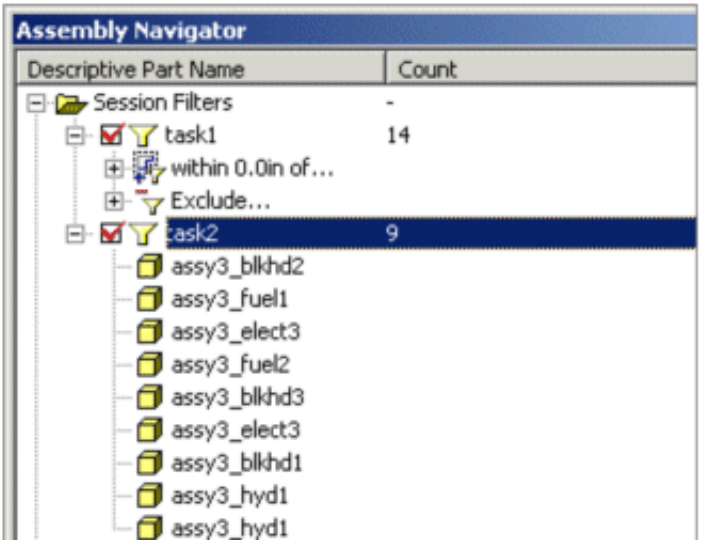
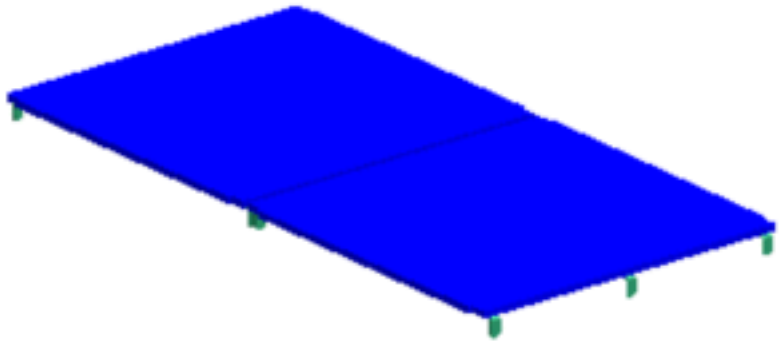
☐ On the task2 filter node, click MB3 and choose Components → Blanking → Isolate



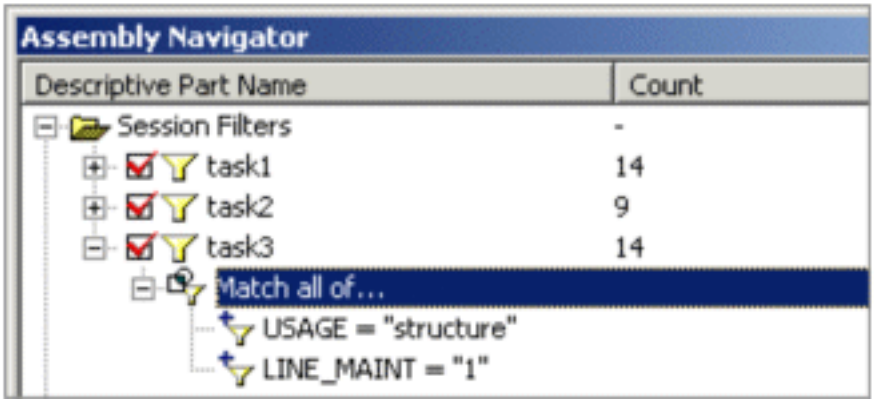
Step 4: Save your bookmark using the same name so it will overwrite the old copy (assy3).

Task 3 Solution:
Assuming the assembly is open and all components are visible:



Step 1: Create a filter that will isolate the desired components.

- ☐ Select the Filters in Session node of the Assembly Navigator.
- ☐ Select the Add Attribute Search to Filter icon from the toolbar.
- ☐ Choose the By Attribute tab.
- ☐ Enter usage = structure and press OK
- ☐ Enter task3 as the new name
- ☐ Reselect the new filter and choose MB3 → Add to Filter ...
- ☐ Choose the By Attribute tab.
- ☐ Enter line_maint = 1 and choose OK.
- ☐ Select the USAGE = "structure" and LINE_MAINT = 1 lines of the new filter.
- ☐ Choose the Match all of... icon on the toolbar.



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Step 2: Test the filter by isolating it's components.

☐ On the Task3 filter node, click MB3 and choose Components → Blanking → Isolate.

Step 3: Save your bookmark again.

Review (Frequently Asked Questions)

Q. How are filters saved?

A. Filters are saved by creating a Bookmark.

Q. At what level of an assembly can the component filters be accessed or used?

A. All filters can be accessed from any level of an assembly as long as the Bookmark was saved and retrieved accordingly.

Q. What is a bounding box? How is it used? How is its size and orientation determined? Do subassemblies have bounding boxes?

A. A bounding box is defined as being the minimum size box shaped container that could contain the component it belongs to. The system may use the bounding box volume in determining whether a given component is in a specified zone or not. Depending on the version of NX the part was last saved in the bounding box may be sized to contain all or part of the solids or sheet bodies within that part. No, subassemblies contain no immediate geometry and so do not have a bounding box.

Q. What zones retain associativity to the "spatial" model size?

A. Only components or the Work Part have spacial associativity when used in a proximity filter.

Q. Can zone definitions be modified?

A. An existing zone can be modified by choosing Component Filter → Define Zone, and selecting the desired zone from the listing. A box zone may be edited by changing box lengths and / or transform. A plane zone may be edited by flipping the plane normal direction and/or transforming the plane.

Q. Can a "spatial" Zone other than a box shape be defined?

A. By choosing Zone → Box, and defining a box with a zero height you can create "bounded" plane , that when used with the Intersects With operator, will limit the amount of data highlighted to only those components that fall within the confines of your "bounded plane" (instead of a true plane zone that would be unbounded in its definition and highlighting). Be aware that because this is still a box zone the operators above and below are not available.

Summary

This lesson concentrated on how to manipulate several assemblies using Bookmarks, Attributes, Filters and Zones.

In this lesson you:

- Used a Bookmark
- Used Filters
- Used Zones
- Create Bookmarks and Filters
- Used Zones to Manipulate Assemblies

Lesson

3 *Representations*

Purpose

This lesson will introduce Representations which create faceted bodies in the work part.

Objectives

Upon completion of this lesson, you will be able to:

- Create and use Representations.
- Recognize advantages and disadvantages of creating Representations.
- Recognize when to change tolerance settings for representation creation.

Representations

When working with large assemblies, often the most time consuming activity is waiting for your assembly to retrieve and display. When using representations, the time required to initially retrieve and display your assembly can be dramatically reduced. Representations are very useful for opening assemblies quickly and visualizing their components. They are also very useful for clearance analysis.

Representations are faceted objects which are associated to the defining solid. By maintaining this association, the representations can be automatically updated whenever the underlying solid changes. A facet is a planar polygon that is sized and oriented to represent a face or portion of a face.

Representations can be created and stored in two different ways.

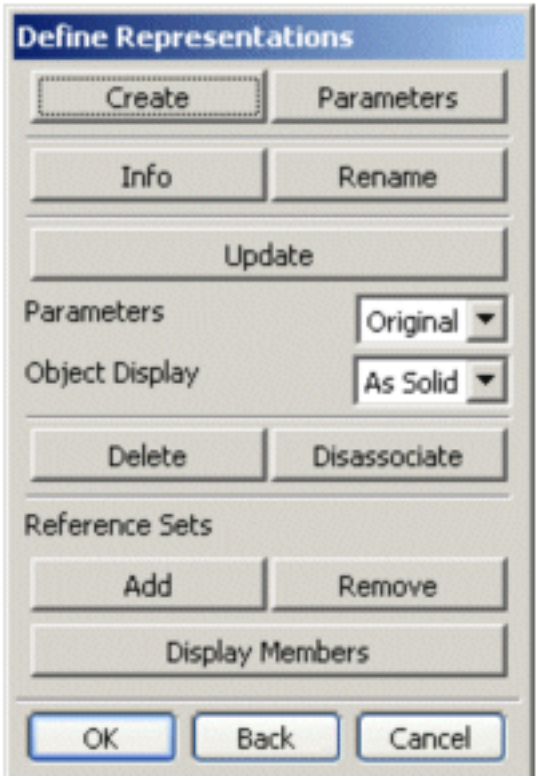
- You can create representations in the detail part containing the defining solid, then add the resulting faceted bodies to a reference set.

- You can create representations in the assembly part which contains the component objects.

Representations will only be visible when the solid body is not visible, either by not being in the current reference set or by being unloaded. The system will not display the representation if the solid is on an invisible layer or is blanked. A representation will display if it has been disassociated from its defining solid.

Define Representations Dialog

The Define Representations Dialog is accessed through Assemblies → Advanced → Representations.



Create: Allows you to select objects using the class selection dialog for faceting. The system will create the facets according to your current Parameters settings. The system will then allow you to select or define the reference set you want the representations to reside in.

A given solid occurrence can have only one representation associated with it within a part file.

Parameters: This option allows you to set the creation parameters for faceting. The parameter settings are stored with a representation so that future updates will be done using the same parameters.

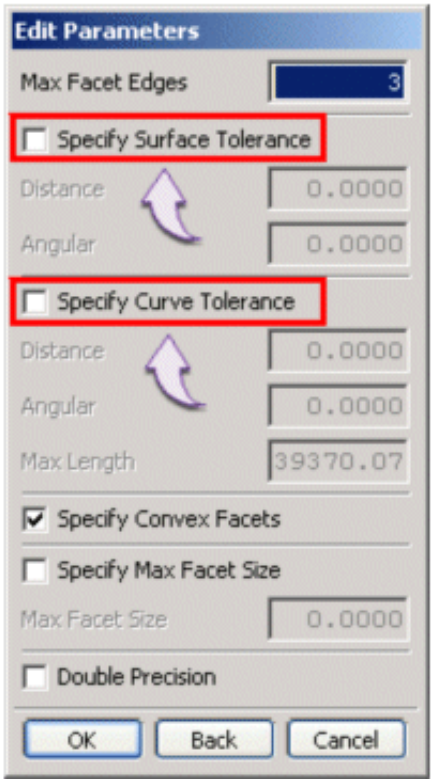
When creating representations for piece parts, you normally will not need to change the default parameter settings.

However, when creating a representation at the assembly level, you have a choice as to how accurate you would like your representation to be. Your choice will affect speed of retrieval, display , and hidden line removal, as well as its usefulness in certain modes of clearance analysis.

- If you leave the tolerance settings alone, your representations will be created using tolerances that were determined for each component individually. The resulting representations will have a higher "resolution"

allowing you to zoom in on them with little graphic distortion; however the performance is slightly decreased.

- By toggling the tolerance switches, the system will determine the best setting for viewing the assembly as a whole. The resulting representations will appear fine when viewed within the entire assembly, but as you zoom in on them you may see reduced resolution of graphics (i.e. circles will begin to appear as octagons etc.)

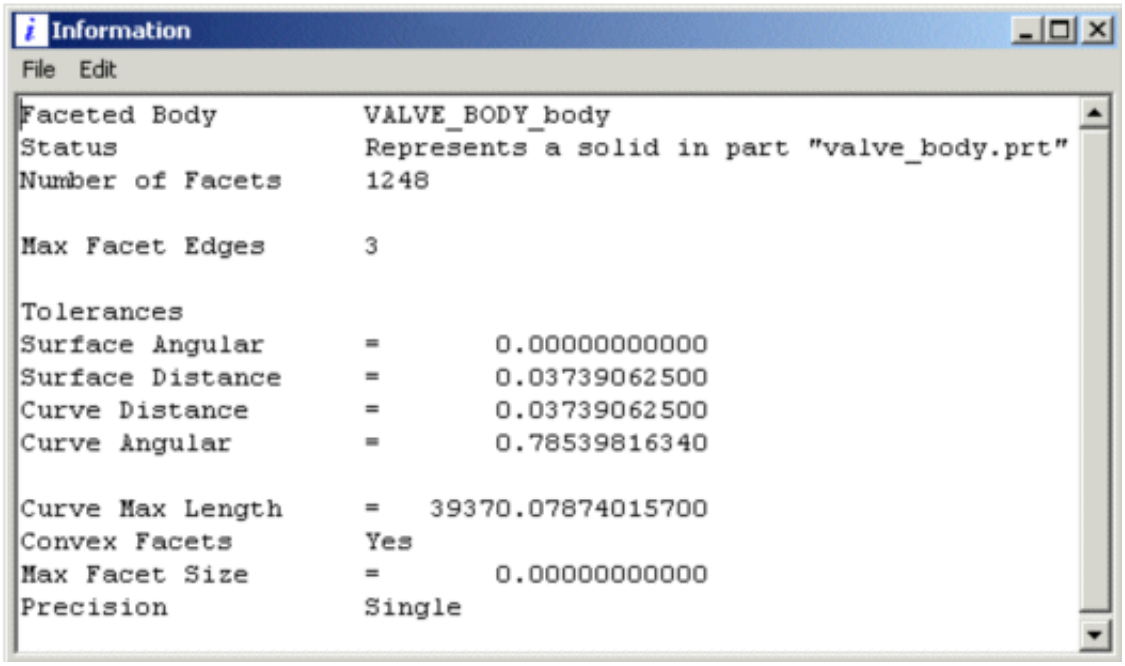


Tip: To ensure the correct tolerances are used, it is a good idea to "FIT" the assembly within your screen before creating your representations.

Another parameter that may need to be changed is " Double Precision". You may want to use double precision if you are doing a Clearance Analysis and you have small components which are a long distance from the origin. In order to get the most accurate results you should toggle the Double Precision option ON. You should be aware that double precision representations take up approximately 30% more memory than single precision and decrease performance.

To obtain further information on these parameters as well as those not discussed, it is recommended you refer to the technical documentation.

Info: Allows you to select a representation and get a listing window displaying information about an existing representation including the part it represents and the tolerance settings used in its creation.



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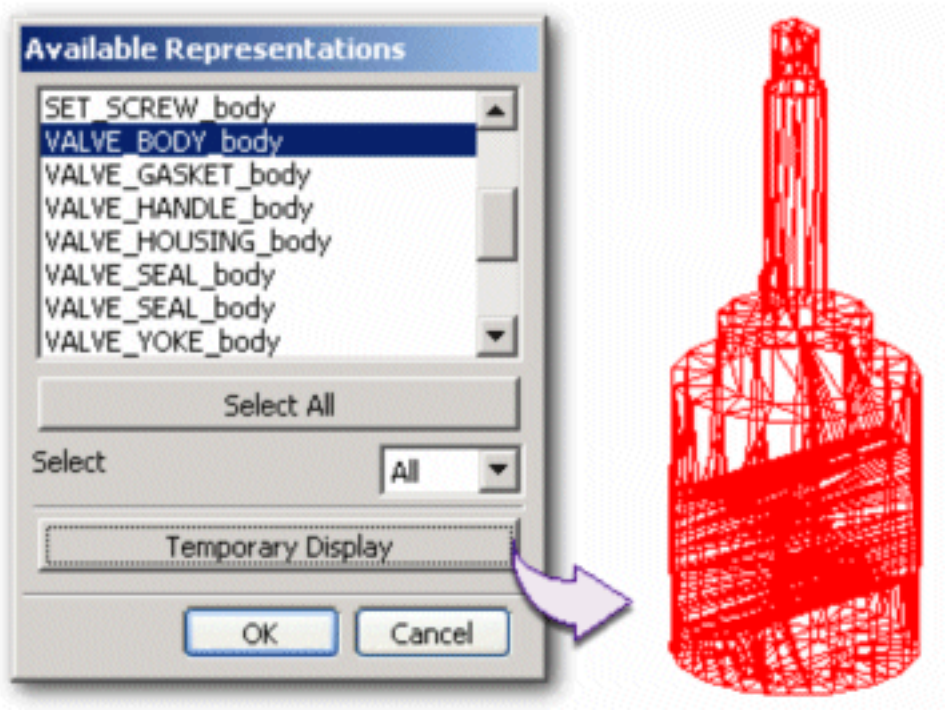
You can also select Temporary Display and the actual facets that make up your representation will appear.



Rename: Allows you to rename an existing representation. The default name for a representation is dependent on the object to which it is associated.

If the representation is of a solid body, the default name will be "body".

If it is associated to a face, the default name will be "face".

If it is associated to a component, the default name will be the component name with "body" added as a suffix. See the above listing window for an example.....VALVE_BODY_body.

Update: If your solid geometry changes, the Representation will automatically update the next time you save the part containing the representation. In some cases, the Representation updates as soon as a modeling update occurs. If the update does not happen automatically you may choose to force an update by choosing the Update option on the dialog. In either case, you must have the defining solid loaded in order for the update to occur. You have the option of having them updated using their Original parameters or the current Default parameter settings.

Object Display, you can choose between As Solid and Current.

- As Solid resets display characteristics (such as the current color and thickness) to that of the owning solid when the display is updated.

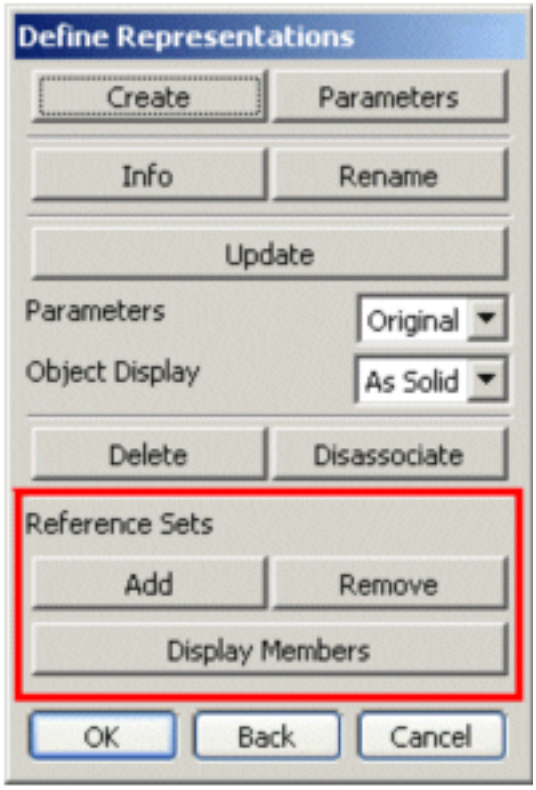
- Current keeps the display characteristics as they were in the faceted representations just before the update.

Delete: Allows you to delete selected representations. This differs from using Edit→Delete in that you can delete hidden representations with this option which you cannot do with Edit→Delete.

If you delete a solid that has an associated representation, the representation objects will also be deleted. The same is also true in the case of removing a solid from an assembly in which representations have been created.

Disassociate: Allows you to break the association with the defining solid. The representation becomes just a faceted object from that point.

A disassociated faceted object can and will display at the same time as a solid.



Add: Allows you to add additional faceted objects to reference sets.

Remove: Allows you to remove faceted objects from reference sets.

Display Members: Allows you to display all the representations that are members of the selected reference set.

Using Representations

You now have to choose where you will create your representations. Your choices for where to create representations are:

Method 1 - In the Mock-up part above the master assembly

Method 2 - In each of the major subassemblies

Method 3 - In each of the piece parts (in a separate reference set)

The method you choose should be based on your requirements and will affect:

(A) In which assemblies you will have access to the representations

(B) How they can be used

(C) Assembly opening time

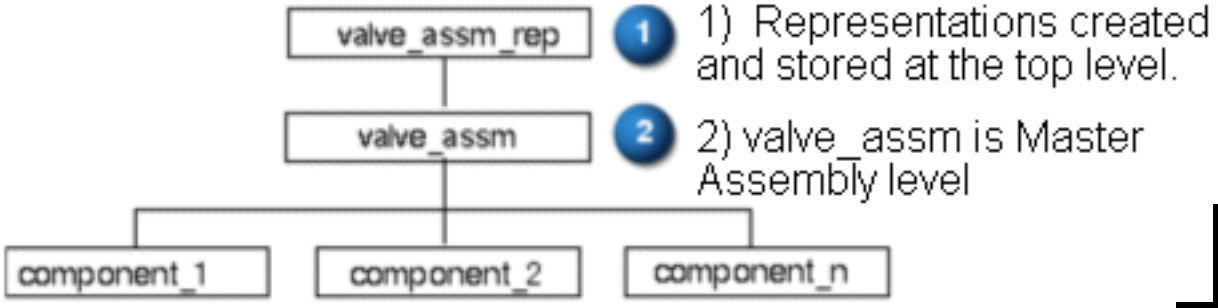
(D) Update of representations

(E) How you control what you will see (facets or solids)

When determining which method you follow, you should consider the following questions:

- Who are the main users of the large assemblies?
- What types of activities will they be involved in?
- Will they be altering the structure of the assemblies (i.e., adding or removing components)?
- Will clearance analysis be used?
- Is the main objective the visualization of the large assembly?
- Are there data creation standards for your company?
- How long can I wait for my assembly to retrieve using solids?

Method 1: Mock-Up Level Representations



(A) The faceted representations are only accessible when the mock-up part is the displayed part. Anyone accessing the assembly at a lower level won't see or load the representations, or be burdened by the memory overhead of carrying the representations.

(B) Main use is for extremely rapid retrieval and clearance analysis. Since components initially are not loaded, most normal component operations are not available.

(C) This yields the fastest load time of any option since all of the data is coming from one file.

(D) To maintain accuracy of your representations you should regularly load all changed components and the mock-up. You can then update the representations and save your mock-up part.

(E) When loading the mock-up part, you will load all representations and you will see them all on your screen unless you adjusted the layer settings to make some layers invisible. Once you have loaded components from your assembly, you can change your displayed part to the master assembly to visualize only loaded parts.

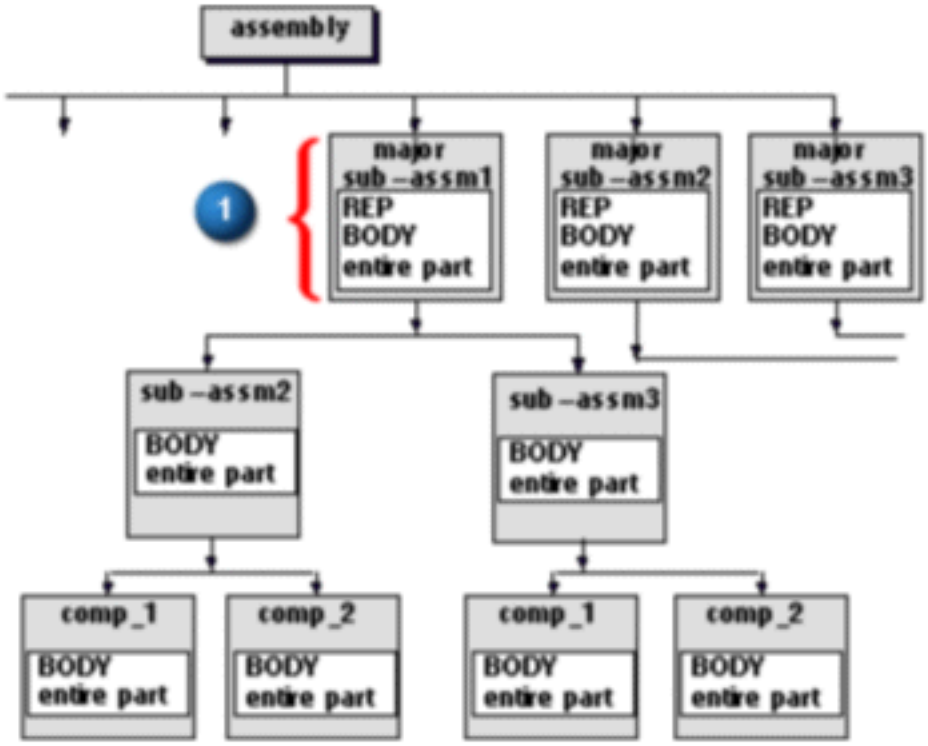
Representations are only seen when the solid geometry is not loaded or is not within the displayed reference set. You can view the Representations by loading an assembly using the load "No Components" option or by using the "Default Reference Set" option and loading the empty reference sets (both settings are found in File→Options→ Load Options). This will make the retrieval and display very fast. You can then work on selected components by either opening them from the Assembly Navigator or by replacing their reference sets depending on the load method chosen for initial retrieval.

The assembly display can be switched back and forth between solids and representations by replacing reference sets between empty (representation displays) and BODY (solid displays). You can also use Open and Close from the Assembly Navigator or Component Filters. An assembly can also mix and match its display, so some components are displayed showing solids and others are displayed with representations.

Tip: Since faceted bodies are visually indistinguishable from solids, it can get confusing trying to determine what is loaded and what is not. A good way to make that distinction is to make use of the partial shading function. This requires that you previously edit the object display and enable partial shading for either your solids or your representations.

This method is good if you often need to see the whole assembly. It has the advantages of allowing a non-owning user to create and use the representations as well as being straight forward and easy to implement. This approach will not be of much benefit to those users who frequently work on assembly subsystems since they would get more data displayed than desired.

Method 2-Major Sub-Assembly Level Representations



(A) Any assembly that contains the major sub-assembly can view the representations for that sub-assembly.

(B) This approach is helpful if you want to see the entire assembly but might concentrate your efforts on one of the major sub-assemblies.

(C) This method is very fast since the operating system need load only a few parts in order to see the entire assembly. You could also use a filter to load only some of the sub-assemblies saving additional time.

(D) This method is similar to Method 1. The representations will require updating via the same type of process discussed for that method.

(E) You will be able to "turn off" the display for various sub-assemblies by closing them or replacing the current reference set with the Empty reference set.

One drawback is that if you are using filters to load components, any parent sub-assembly will be loaded and you will see all of the representations for that parent sub-assembly.

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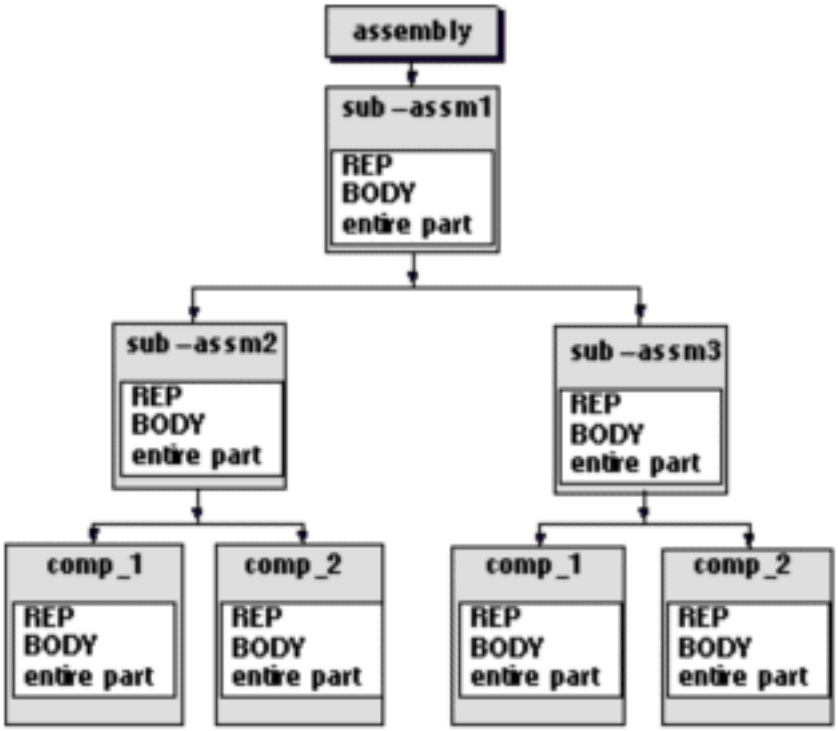
Method 3-Piece Part Level Representations



(A) Any assembly that contains the component can view the representations for that component.

(B) This method can be used for most any operation except solid based activities, most notably mating conditions and promotions. For example, you might use filters to select which components you want to load, and identify components for Edit Structure operations and Exploded View operations.

(C) Time required to load the assembly is significantly reduced compared to loading solids, but is still much slower than the previous methods discussed. This is because the operating system must still locate each component and load the faceted data from it.

(D) Updates of your representations is almost a non-issue since each time you change and save the component the representation will be automatically updated.

(E) You can control whether you are seeing the solid data or the representations by replacing reference sets as well as opening and closing components.

This method may be the most commonly used. It is relatively easy for users to understand and use. The main drawbacks are decreased performance and that everyone has to remember to create the representations for all parts he or she makes.

The above methods can be mixed and matched to suit the needs of the users who will need to retrieve the model. For example, you may decide you want representations defined at the detail level and also at the major subassembly level. This would offer many of the benefits of both worlds but would add a level of confusion.

The following table summarizes the methods and repercussions of representation creation:

	Method 1 (Mock-Up)	Method 2 *2* (Sub-Assembly)	Method 3 (Piece Part)
A	Accessible only if the displayed part is the mock-up. All or Nothing.	Accessible by any assembly to which it belongs.	Accessible by any referencing assembly.
B	Fast loading and visualization, clearance analysis.	Quick visualization of entire assembly and easy transition to working on subassemblies	Increased retrieval performance and most component operations available.
C	Fastest load time option.	Very fast load time	Fast load time compared with solids retrieval.
D	Requires regular update by loading the changed components and updating.	Similar to Method 1.	Updates occur automatically when editing your components and saving them.
E	Use partial shading, or Assy Nav. tree method, or change displayed part to master assembly level, or layer settings.	Turn off display by closing subassembly or by replacing reference sets to the empty set.	Control of visible representations is controlled with the use of reference sets.
F	Cannot explode representations	Can explode each sub-assembly component	Each representation in a component explodes.
G	Clearance analysis won't show names	Sub-Assembly name will appear	Shows parent component name.

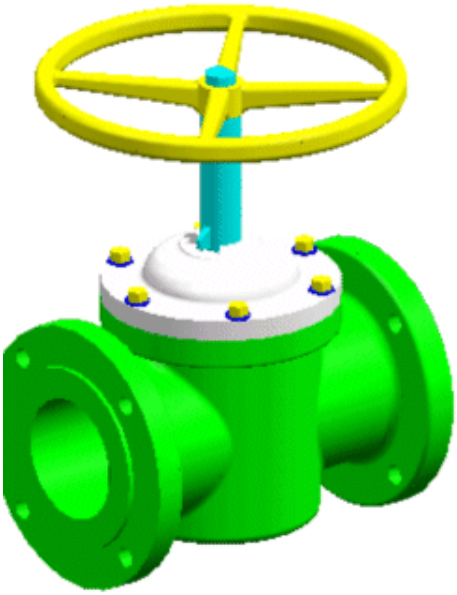
Additional Application: Simplified Display Representations

If your assemblies contain complex parts (those with many features), you may consider first suppressing some of the features in the underlying solid, creating the representation, then disassociating it. Disassociation is required since upon unsuppressing your features in the solids, the representations would update to reflect that change. Now you can swap reference sets and see the simplified version or the full blown solid.

Since the faceted objects are no longer associated to the solids, steps should be taken to ensure that the facets are recreated at appropriate intervals. This may be worthwhile if you have complex parts with small features that take a long time to display

Activity: Valve Assembly Representations

In this activity, you will be creating representations for the components of the Valve Assembly using Method 1 described previously.



Step 1: Create your application specific part.

☐ Create a new part called valve_assm_rep using inch units in the act4 sub-directory.

☐ Add the part valve_assm, found in the act4 directory, to it as a component.

Step 2: Replace view in layout to the TFR-TRI view and "Fit" the geometry to your display.

Step 3: Set your parameters for your Representations

☐ Select Assemblies→ Representations, then select the Parameters button.

☐ Toggle the tolerance switches ON for "Specify Surface Tolerance" and "Specify Curve Tolerance".

The system will evaluate your geometry and set your parameters appropriately.

☐ Select OK to accept the parameter settings and to dismiss the dialog.

Step 4: Create your Representations

☐ Select the Create button.

☐ Choose Select All from the class selection menu

☐ Choose OK to accept

The next menu will ask you for the reference set name in which to hold the representations. If the reference set does not exist, you can type the name in. In this case since we are at a top level assembly, a reference set is not required. If you select Cancel without typing in a name, the Representations will not be added to any reference set.

Step 5: Select Cancel. You now have representations in your part file.

Now you will modify your Representations so they have partial shading capability. This will help us distinguish which parts on the screen are loaded.

Step 6: Enable Partial Shading for the newly created representations.

Partial Shading is only available if your work station is logged in using a 3D driver.

☐ Choose Edit → Object Display

☐ Set your Type filter option to Faceted Body.

☐ Choose Select All.

☐ Notice the status line tells us that no objects are selected, Why?

Even though you just created your faceted representations, they are not available for display or selection because their associated solids are being displayed.

☐ Choose Cancel in the Class Selection dialog box.

☐ Open your Assembly Navigator and replace reference set to Empty for the valve_assm.

Notice the geometry disappears then reappears. You should now be looking at your representations. The geometry does not appear any different. This could become confusing if you had some of the components loaded and others that are not. Let's try again to enable partial shading.

☐ Choose Edit → Object Display.

☐ Set your Type filter option to Faceted Body.

☐ Choose Select All.

- ☐ Choose OK.
- ☐ Set the Partial Shading Option to Yes
- ☐ Choose OK
- ☐ Replace Reference set back to Entire Part.

Step 7: Save your assembly, then close all parts.

On Your Own

Now to test and become more familiar with how representations handle design changes try:

- removing a component from your assembly.
- adding a new component to your assembly.

In both cases check whether or not the representation is still accurate, then correct if required.

Representations and Drawings

While representations are not currently designed to be used with drawings there are some applications in which they can be very useful.

If you add a view containing representations to a drawing, that view will not have the Hidden Line Removal (HLR) performed on it. However, you can create an HLR rendering using Representations imported onto your drawing. This view will not be associative to your solid geometry and therefore will need to be managed manually.

To create such a view:

Choose View→ Visualization→ Assembly Hidden Line.

This operation will use faceted representations to quickly create an HLR rendering. If representations are not available, the system will first create temporary faceted representations. Upon completion of this operation, the system offers you the option of saving the HLR into a new file.

ChooseSave and then enter a name for your HLR rendering, e.g. temp.prt.

This new file can then be imported into your drawing file and then onto your drawing.

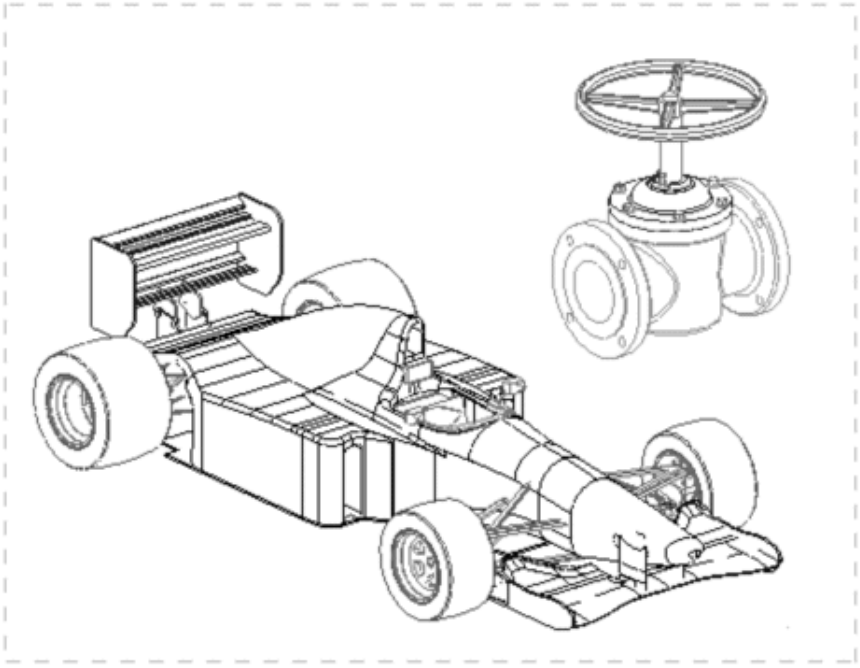
File→ Import→Part→ then select your part temp.prt.

Now you can create a view and add it to your drawing.

Obviously this is not the method you will want to use for most of your drawings. However, for very large assemblies that take considerable time to generate drawings, this can be a big time saver. Since the resulting drawing is non-associative, the user will have to recreate the drawing when updates are needed.

Activity: Valve Assembly Representations (optional)

Create a drawing containing the Benetton car and the valve assembly. Hint: The drawing shown was done on a "E" size sheet, the car view is 1/4 scale.



Step 1: Open the car assembly using Representations.

☐ Set your load options to load No Components.

☐ Open "*demo_mockup*" from the car directory.

Step 2: Generate an HLR rendering of the car.

☐ Choose View→Visualization→Assembly→Hidden Line

☐ Choose Save, then enter ***_hlr.prt in your t# directory where *** denotes your initials. Then choose OK.

☐ Choose Cancel.

Step 3: Import the HLR part into your demo_mockup file onto its own layer.

☐ Make your work layer 19.

☐ Choose File→Import→Part and then OK the options menu.

☐ Choose your HLR part.

☐ Choose OK twice.

Step 4: Create a view of that geometry.

☐ Make all layers invisible leaving just 19 visible.

☐ View→ Operation→ Save As → enter a name for your view.

Step 5: Add that view onto your drawing.

Step 6: Repeat the above steps to create an HLR rendering for the valve_assm but import the geometry into the car file.

Step 7: Close all parts.

Representations Created At The Piece Part Level

The vise assembly, which many of you have worked with in the standard Assemblies Implementation class, is also available. This assembly has had representations created and added to "REP" reference sets at the component level. To further your understanding of how working with an assembly set up in this manner is different, you should experiment with this assembly. This assembly can be found in the act4 directory. Suggested steps are:

- Retrieve the assembly using the Load Options set to load No Components.
- Retrieve the assembly using REP as the default reference set.
- Load part of the assembly using a filter you create.
- Create a filter to open only the fasteners in the vise assembly.
- Experiment with the Assemblies→Edit Structure menu to see what component operations are available.

Summary

This lesson introduced Representations which create faceted bodies in the work part.

In this lesson you:

- Created Representations
- Used Representations

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Lesson

4 Wrap Assembly and Linked Exterior Feature

Purpose

This lesson will introduce you to wrapping assemblies, editing wrapped assemblies and how to develop linked exterior features.

Objectives

Upon completion of this lesson, you will be able to:

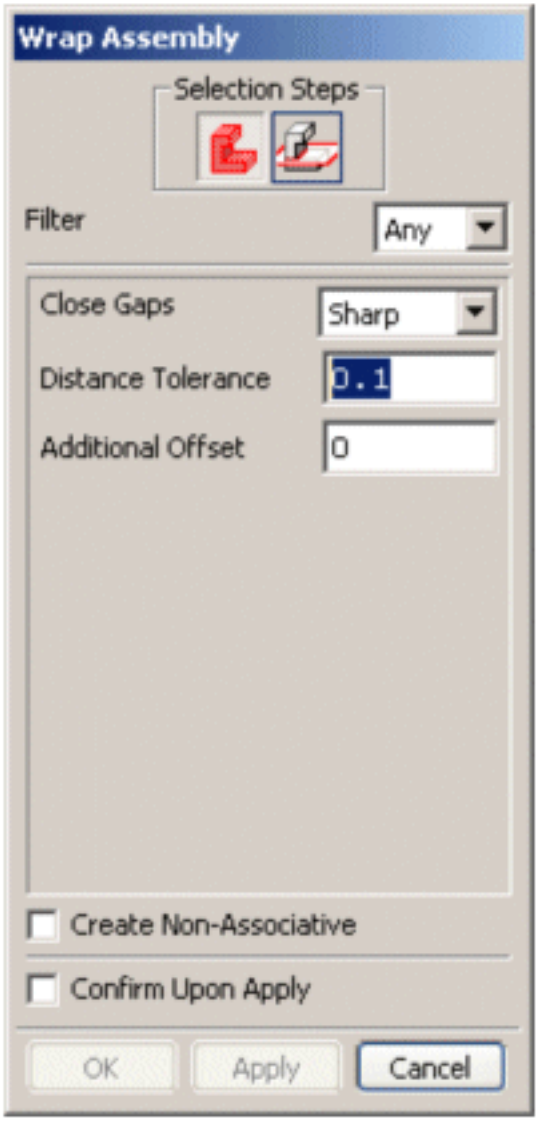
- Create wrapped assembly geometry and understand how to do isolated edits of the wrapped assembly.
- Develop a linked exterior feature

Wrap Assembly Overview

The Wrap Assembly function (also known as assembly envelope) simplifies a complex assembly by computing a solid envelope that encloses the assembly. This effectively "shrink wraps" the assembly with a convex polyhedron of planar faces.

This feature is similar to the Wrap Geometry feature that is described in the Modeling Help, except that the WRAP_ASSEMBLY feature allows interpart links in the input geometry. It is created in the context of the displayed part, but resides in the work part. Input data can reside anywhere in the current assembly, because interpart links relate the data to the WRAP_ASSEMBLY feature.

When you choose Assemblies→ Advanced→ Wrap Assembly, the Wrap Assembly dialog appears.



Wrap Assembly Dialog Options

Geometry to Wrap: Selects any number of components or geometry (solids, sheets, curves, or points) in the assembly to be wrapped. Since the result is a solid body, the input should not be coplanar. (Components will be selected if Select Features or Select Components is toggled ON, while objects will be selected if Select General Objects is toggled ON. These options are available on the Global Selection toolbar and Edit→ Selection.)

Filter: Limits the types of objects that are selectable.

Close Gaps: Determines how to close gaps between offset faces.

- **Sharp:** Each planar face is extended until it meets adjacent faces.
- **Beveled**: Planar faces are added in gaps to create a beveled effect. To avoid creating tiny faces, bevels will not be narrower than the Distance Tolerance. Gaps smaller than this are closed using sharp edges.
- **No Offset**: Faces are not offset. This is faster, but the result usually does not enclose original data.

Distance Tolerance: Specifies how detailed the result should be.

Additional Offset: Applies an additional offset to the faces of the resulting body. This option is grayed out if the Close Gaps option is No Offset.

Create Non-Associative: Creates an unparameterized feature.

Confirm Upon Apply: Preview what the results will look like. An additional dialog will appear after you choose Apply that lets you accept or reject the results, or you can choose one of the analysis options (i.e., Interference, Examine Geometry, Curve Analysis, Face Analysis, Deviation, and Section Analysis). See the Gateway Help for more information about the analysis options.

Splitting Planes: (Selection Step- Optional) Logically subdivides the model into smaller pieces so the envelope is more precise. The geometry you select is split at the planes. Geometry on each side of a splitting plane is wrapped separately and then reunited into a single solid. Since each side of the plane is wrapped separately, the set of geometry on each side must not be coplanar.

- Each plane should make contact with and split at least one piece of geometry. This guarantees that the final set of bodies will contact one another and can be united into a single result. Otherwise, an error is issued.
- If multiple non-parallel planes are given, the planes will subdivide each other. A plane which does not split any data within a subdivision is ignored. Splitting planes are optional.

- To define a splitting plane, select a datum plane from the graphics window, or choose Define to bring up the Plane Constructor (see the Gateway Help for more information).

- New planes are numbered and added to the Defined Planes list box. They are also displayed as numbered temporary planes in the graphics window.

- You can remove one or more planes by selecting them in the Defined Planes list box and choosing Remove. When a plane is removed, the list is reordered, and the remaining planes are redisplayed. You can also use Redisplay Planes to regenerate the temporary display of the planes.

- Datum planes can be selected from anywhere in the assembly. Input for plane smart objects created using Define is restricted to the work part.

Split Offset: Applies this positive offset to each side of the splitting plane. Effectively, each plane becomes two overlapping splitting planes, guaranteeing that the result on each side of the splitting will overlap and will reunite without a non-manifold condition. This is useful when the data on each side of the plane meets at a single point.

Defined Planes list box: Lists the planes that have been defined.

Define: Lets you define a new plane by bringing up the Plane Constructor. See the Gateway Help for more information about the Plane Constructor.

Remove: To remove a plane, select it in the Defined Planes list box and choose Remove.

Redisplay Planes: Regenerates the temporary display.

Wrap Assembly Procedure

1. Use the Geometry to Wrap selection step to select objects to wrap.

2. If an offset is to be applied to faces when calculating the envelope, select a Close Gaps method (Sharp, Beveled, or No Offset) for closing gaps that may result.

3. Specify a Distance Tolerance to determine how detailed you want the result to be. The value is used to generate points from the input data. The points are then used to calculate the envelope. For curves, this value equates to the maximum chordal deviation. For bodies, this equates to the maximum facet to surface deviation. This will default to one hundred times the part's distance tolerance.

4. Specify the amount of Additional Offset to be applied.

5. (Optional) Use the Splitting Planes selection step to define planes used to split the geometry, to better define the envelope's shape.

6. (Optional) Toggle Create Non-Associative ON if you want to create an unparameterized feature.

7. Toggle Confirm Upon Apply ON if you want to preview the results.

8. Click OK or Apply to create the WRAP_ASSEMBLY feature.

Wrap Assembly Notes:

The WRAP_ASSEMBLY feature is associative (unless you chose the Create Non-Associative option), and will update when changes are made to the input geometry.

In order to create an associative feature, the Assemblies_AllowInterPart customer default must be set. Otherwise, when you open the Wrap Geometries dialog, you will get a message that links created in the session will be non-associative. (This is also true for the WAVE Geometry Linker.)

When Delay Interpart Updates (on the WAVE Associativity Manager) is toggled ON, component changes are not propagated to the WRAP_ASSEMBLY feature until you update the assembly.

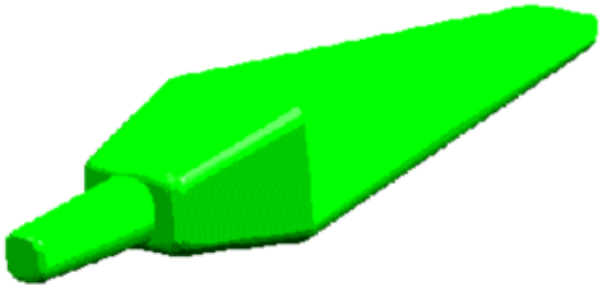
Updating the feature when any interpart links are broken causes the feature to become "dumb". The feature uses the body from the last successful update, and does not reflect any current changes made to the input geometry. If the link was not broken deliberately (e.g., with the WAVE Part Link Browser), an update warning will be issued in the Edit During Update dialog (see the Modeling Help for more information about this dialog).

Editing Wrap Assembly Features:

For a Wrap Assembly feature, the edit dialog looks the same as the Wrap Assembly dialog used to create the feature, except that Create Non-Associative and Confirm Upon Apply have been removed. The feature resides in the work part, but the feature's original selections were made in the context of the displayed part. If the displayed part has changed, it may be impossible to relate some of the original selections to the current displayed part. Selections that cannot be resolved are lost, and a warning message is issued.

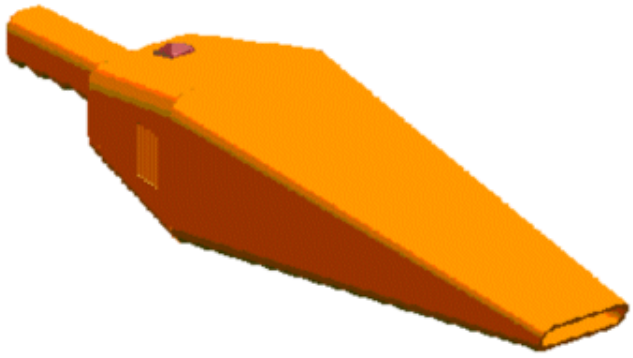
Activity: Wrap Assembly

In this activity, you will be creating a Wrap enclosure for a hand held dust vacuum.



Step 1: Open the Dust Vac assembly and load all of the components

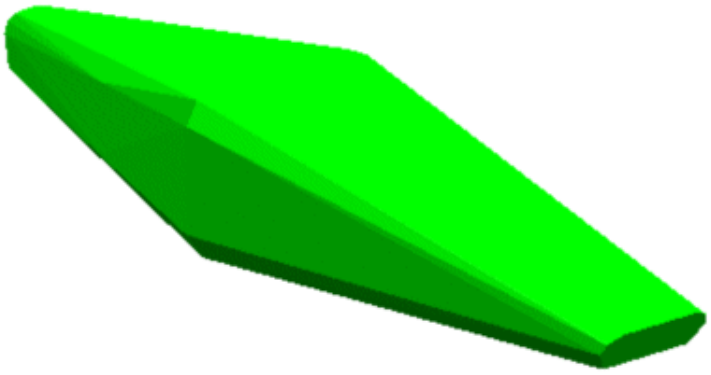
- ☐ Start the Assemblies Application (if necessary).
- ☐ Set your Load Options to From Directory and your default Reference Sets... to Body and Load All Components (if necessary).
- ☐ From the vac folder open the dust_vac.prt.
- ☐ Shade the assembly



Step 2: Create a Wrap for all of the components in the assembly using the default settings

- ☐ Assemblies → Advanced→ Wrap Assembly.
- ☐ With the Geometry to Wrap selection step icon selected, drag a rectangle around all of the components.

☐ Toggle ON Confirm upon Apply and select Apply.



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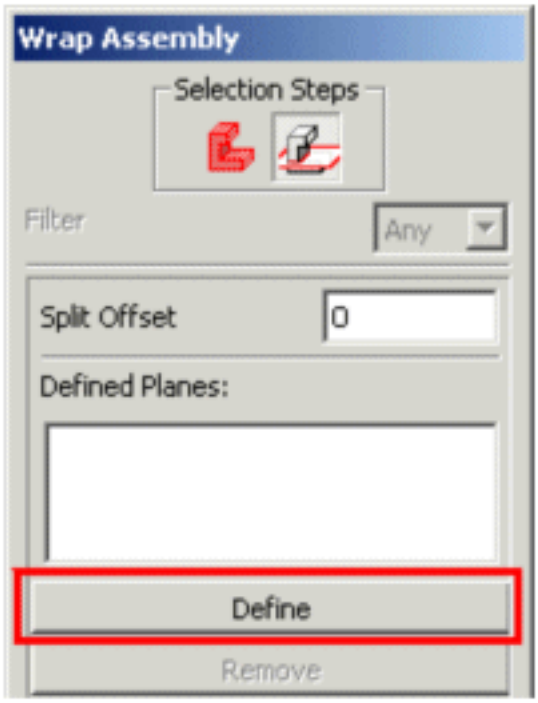
The wrap was successful but it does not define the shape of the handle very well. You will not accept the results and define a Splitting Plane to divide the Wrap.

Step 3: Reject the result and define a Splitting Plane where the handle meets the body

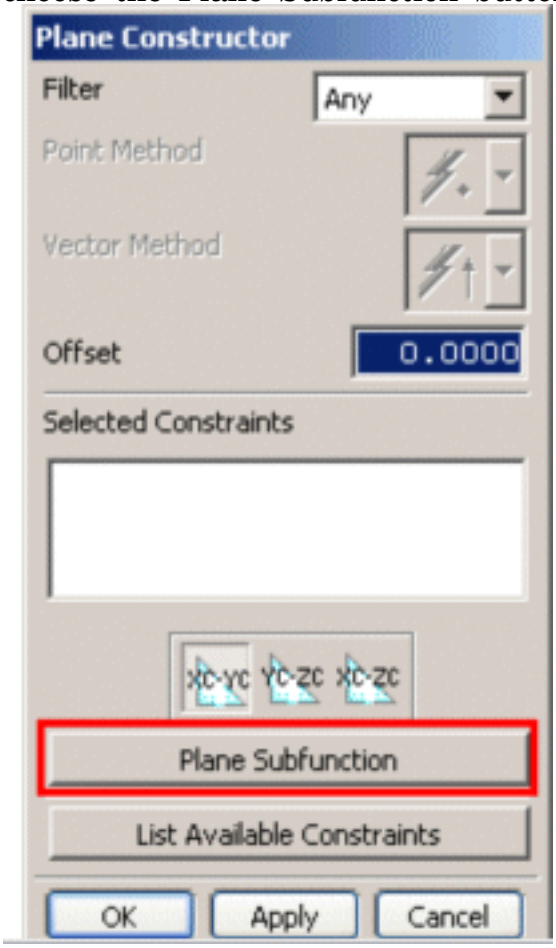
☐ Select Back, on the Confirm Upon Apply dialog

☐ Set the view to Hidden Lines Invisible and rotate it so that you can see where the handle meets the housing

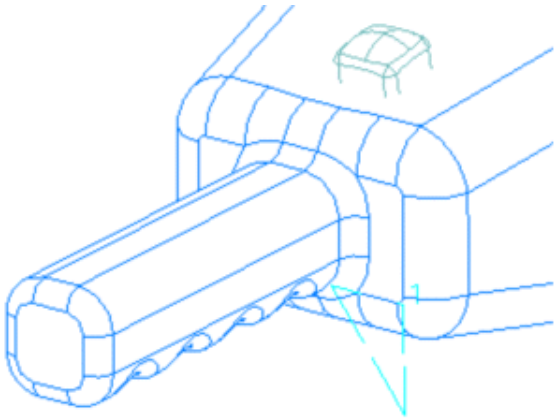
☐ Choose the Splitting Planes icon in the selection step area and choose the Define button.



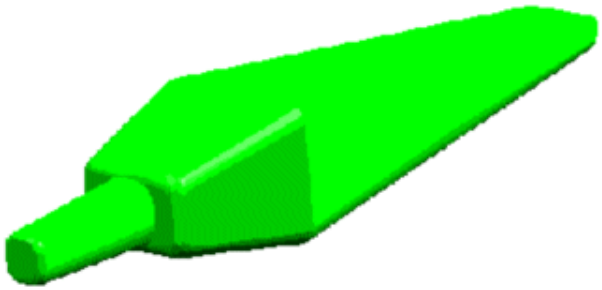
☐ In the Plane Constructor dialog, choose the Plane Subfunction button



☐ Choose the Two Lnes method and then select any two intersecting lines on the planar face where the handle meets the housing.



☐ Choose Apply to see the results. Shade and Fit the Assembly



The new Wrap is more representative of the shape but the facet size is a little large to give a good approximation of the shape. Reject the Wrap once again to reduce the facet size.

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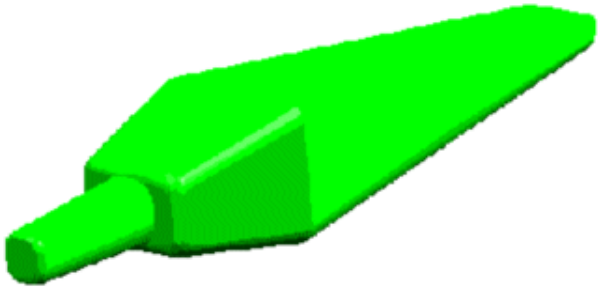
Step 4: Reject the result and decrease the Distance Tolerance to produce more facets

☐ Choose Back on the Confirm Upon Apply dialog

☐ Select the Geometry to Wrap selection icon.

☐ In the Distance Tolerance widget, change the value of 2.54 to *0.254* (ten times greater accuracy).

☐ Choose Apply for the system to recalculate the Wrap.



The Wrap looks much smoother. The visual benefit is obvious but you also need to consider that the file size has just grown because you now are viewing approximately 10x the number of facets more than the original Wrap. Your goal should not be to make the shape look exactly like the solid but it should be a reasonable approximation.

Step 5: Accept the Wrap then Close all parts

Linked Exterior Overview

The linked exterior function allows you to extract faces of selected bodies in an assembly. These can be associative and grouped if desired. Linked exteriors can be used anytime you want to represent an exterior envelop that is to be used in another, higher level assembly. This is especially useful when you must coordinate development with other design disciplines but only spatial envelops are needed for reference and additional assembly data would be superfluous. When you create a linked exterior a new feature, a LINKED_EXTERIOR feature is created.

Activity: Linked Exterior

In this activity, you will create a Linked Exterior feature that includes all of the exterior faces of a garage door opener remote.

Setting up the Model

Step 1: Open remote_assm.prt from the remote subdirectory.

☐ Check your load options to ensure that the Load Method is From Directory, and Load Method is set to All Components (if necessary).

☐ Use File→Open, navigate to the remote subdirectory and select the remote_assm.prt, then choose OK.

☐ Choose the Modeling icon from the Application toolbar or choose Application→Modeling (if necessary).

☐ Choose the Modeling icon from the Application toolbar or choose Application→Assemblies (if necessary).

Step 2: Change the Work layer to Layer 15 where the Linked Feature will be created.

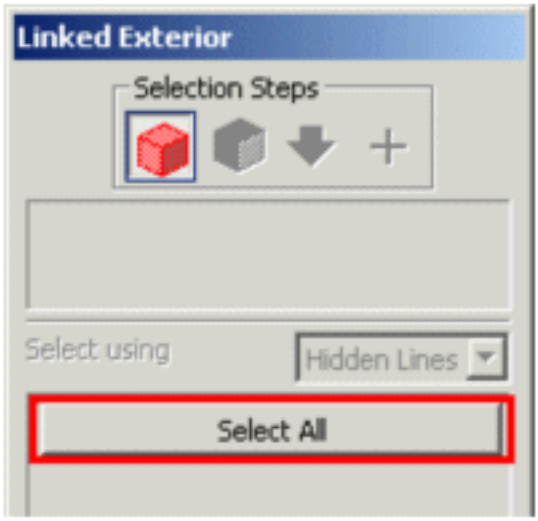
☐ Enter 15 in the Work Layer field on the Utility toolbar or choose Format→Layer Settings, then Work = 15, then OK.

Creating the Linked Exterior

Step 1: Create the Linked Exterior feature.

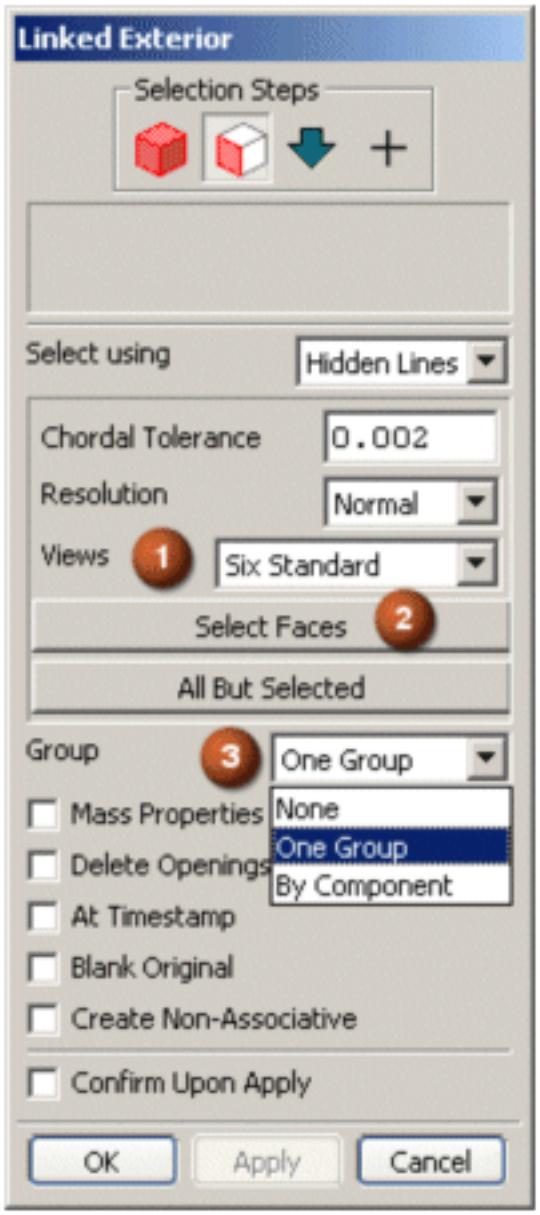
☐ From the Assemblies→Advanced→Linked Exterior

☐ With the Candidate Bodies selection step icon selected, choose the Select All button to identify all of the solids as candidates for linking.



☐ Choose the Exterior Faces icon to advance to the second selection step.

☐ Confirm that the Views is set to Six Standard (1).



☐ Choose Select Faces (2) to start the automatic face selection algorithm.

☐ Change the Group option to One Group (3).

☐ Choose OK twice, accepting the rest of the defaults and initiate the creation of the feature.

The linked exterior has been created.

In the customer default files, you can specify the name of a reference set that will be automatically created and will add the linked envelope automatically. Assemblies_SimplifiedReferenceSet: ENVELOPE

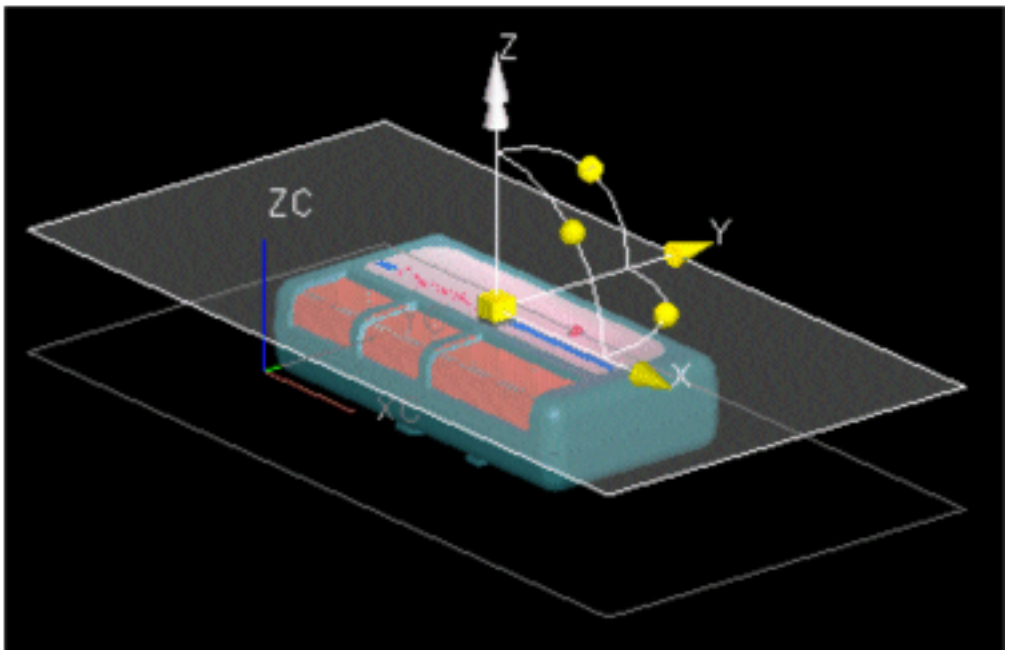
Checking the Linked Exterior

Step 1: Blank all of the components to view the new feature.

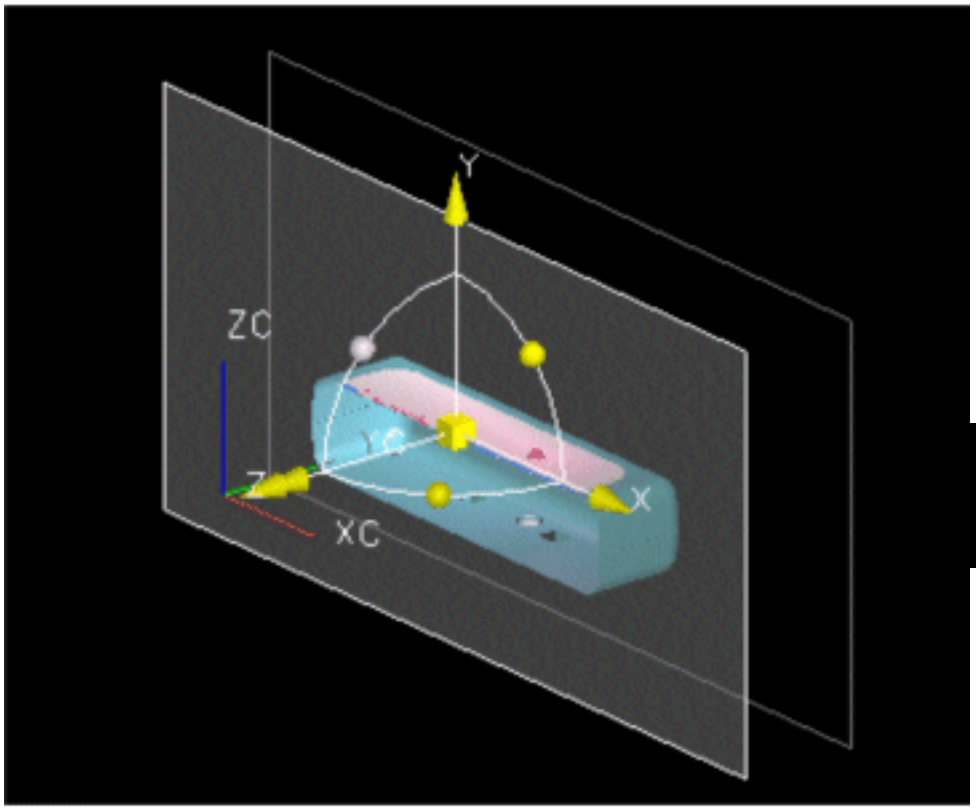
☐ In the assembly navigator, toggle the checkmark in front of each component.

Step 2: Check the linked exterior by doing showing a section cut through the exterior

☐ Choose View→Operation→Section.



☐ Using the Y-Z rotation ball, rotate X-Y plane 90 degrees.



Note that the linked exterior shows just the outside surface of the remote.

☐ OK the dialog after checking the linked exterior.

☐ Choose the Model Navigator tab. (Unix users, choose the Model Navigator icon on the Navigators toolbar.)

Note the addition of the LINKED_EXTERIOR feature.

Step 3: Close all parts.

Summary

In this lesson you:

- Created a Wrap Assembly
- Created Linked Exterior features
- Realized the intent and where they should be applied.

Lesson

5 Assembly Clearance

Purpose

This lesson will allow you to determine the interferences of all components in one operation.

Objective

Upon completion of this lesson, you will be able to:

- Use the different modes of interference.
- Use different analysis modes.
- Perform a Clearance Analysis and interpret the results.
- Execute a batch Clearance Analysis.

Overview

Using NX, you can conduct a clearance analysis on all or a portion of your assembly. NX allows you to perform this analysis either interactively or in batch mode.
There are two approaches to clearance analysis: Interference mode and Clearance Zone mode. You can perform a clearance zone or interference check between all or some of the components in an assembly. This check can be done both interactively or in batch mode. Clearance analysis does not take into account any motion a part may have, only static conditions are evaluated. The system allows the user to input a clearance zone for all or selected components, which it uses to compare against the minimum distance calculated between the bodies. If the distance calculated is less than the clearance zone, an interference is reported. There are three states of interference that the system recognizes:

1. Soft Interference - two objects are situated such that the minimum distance between the bodies is less than the clearance zone, but the two objects do not touch. The system can create a line representing the minimum distance found between the two objects where the interference was detected.

2. Touching Interference - Two objects are touching but do not interfere. When this type of interference is detected, the system will create a point at the location the touching interference was detected.

3. Hard Interference - When two objects intersect with each other, the system can create an interference solid representing the interference.

The Clearance Set

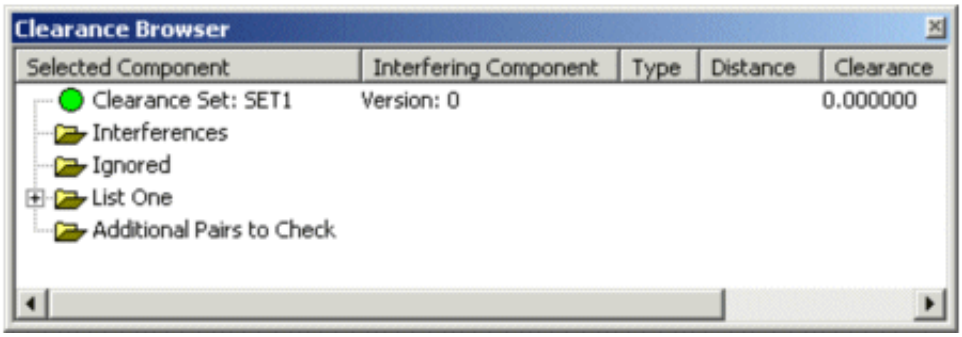
Whenever you do a clearance analysis, the results are stored in a clearance set. This clearance set is stored in the part file from which it was run. Clearance sets are named so you can do multiple analyses between different sets of objects within the same part file and have each run named differently.

Clearance Sets can be created in two ways:

- Using Analysis→Assembly Clearance→Clearance Set→New.
- In the Clearance Browser, using MB3→Clearance Set→New

The Clearance Browser

With the Clearance Browser, you can access other functionality within NX to further analyze any clearance interferences. The Clearance Browser is accessed by selecting Analysis→Assembly Clearance→Clearance Browser.



The Clearance Browser presents important information relating to any clearance set that you may define.

Selected Component	Interfering ...	Type	Distance	Clearance
Clearance Set: TEST_RUN_1	Version: 1			0.100000
Interferences				
3_4 (762)	3_t22 (6)	New (Soft)	0.000000	0.250000
3_3 (367)	3_17 (535)	New (Hard)	0.000000	0.250000
3_3 (367)	3_t22 (6)	New (Soft)	0.000000	0.250000
3_17 (535)	3_8 (546)	New (Hard)	0.000000	0.250000
3_1 (24)	3_t22 (6)	New (Soft)	0.000000	0.250000
3_9 (681)	3_8 (546)	New (Hard)	0.000000	0.250000

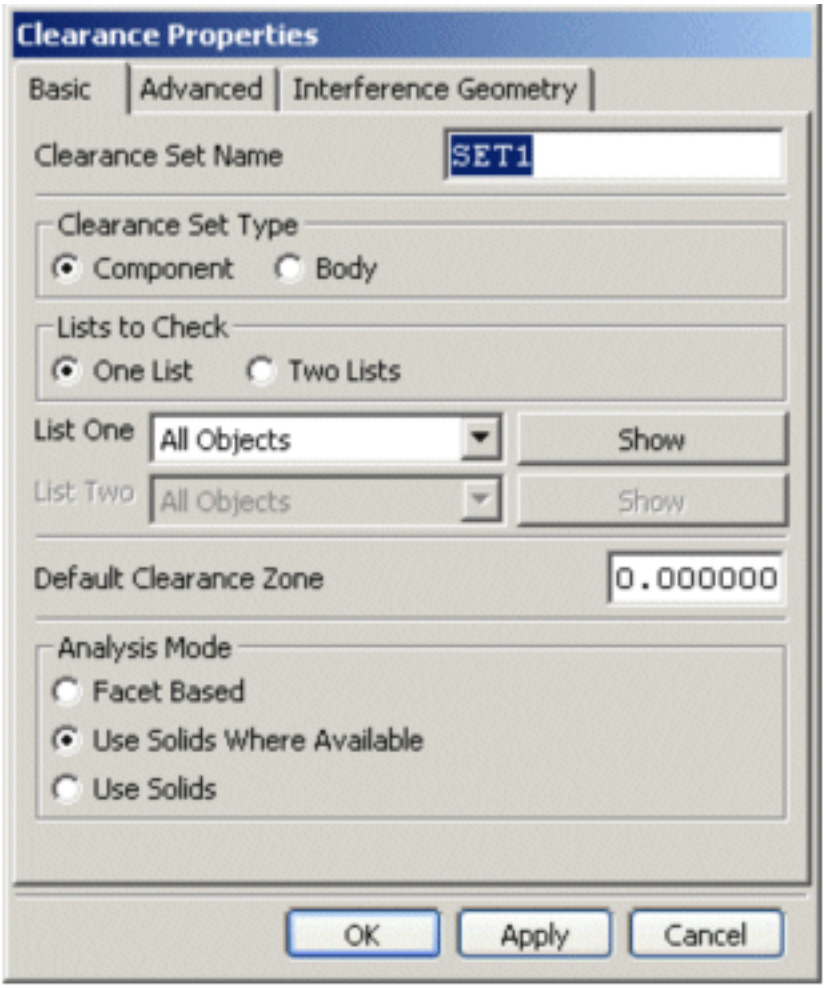
Note the interference symbols in the Clearance Browser for the different types of interferences.

Clearance data, along with important conditions relating to that data, is displayed in the various columns of the Clearance Browser.

Column	Description
Selected Component	Column shows the name and I.D. of the first component of any pair of objects that have been analyzed.
Interfering Component	Column shows the name and id of the second component of any pair of objects that have been analyzed.
Type	Column reflects type of interference: new/existing; soft, touching or hard. There is also an iconic depiction of the type of interference next to the selected component: soft, hard, touching.
Distance	Enumerates the minimum distance between components being analyzed, if known.
Clearance	Reflects any clearance zone that has been specified.
Identifier	A reference number for identification.
Out of Date	Reflects any out of date comparisons based on the last clearance analysis run.
Unloaded Objects	Indicates if component(s) are loaded or not.
Text	Any text that you may want to add.

Clearance Properties

The main dialog for specifying different aspects of your clearance analysis is the Clearance Properties dialog. The information you designate in this dialog is part of your Clearance Set definition.



This dialog can be accessed in three ways:

- Using Analysis→Assembly Clearance→Clearance Set→Properties.
- In the Clearance Browser, by placing the cursor over the Clearance Set node then using MB3→Properties (existing clearance sets)
- In the Clearance Browser, by placing the cursor over the Clearance Set node then using MB3→Clearance Set→New (during specification of a new clearance set)

The Basic Tab

In the Basic page, you can specify:

- the Clearance Set Name
- the Clearance Set Type (components or bodies)
- the number of Lists to Check
- the make-up of the list(s)
- a default clearance zone
- the analysis mode

Clearance Set Name

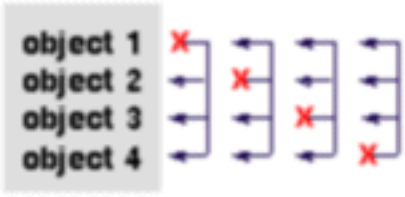
The Clearance Set Name field is where you designate the name of the clearance set for a particular analysis run. Any entries in this field are case insensitive; all entries will default to upper case. When creating a new clearance set, the system will start with a default name of SET1 and will increment automatically as new default clearance sets are designated. In cases where a specific name is designated and that name has already been used, the system will increment that name by 1 until a unique name is determined.

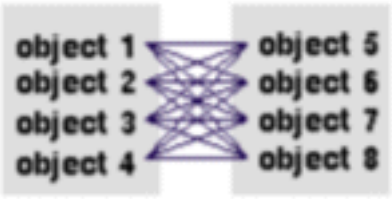
Clearance Set Type

In the Clearance Set Type area of the dialog, you specify whether your analysis will deal with components or bodies. The default for new clearance sets is components.

Lists to Check

Clearance Analysis allows you to have your data evaluated in one of two ways:





1. One List: the system checks for interference between all parts on that list.

2. Two Lists: the system checks all parts on list one against all parts on list two.

A possible application for two lists would be: As your assembly changes you add any new or changed components to list 2 and keep everything else on list 1. This way every time you add a component or make a geometrical change you can double check yourself for any introduced interferences. When you want to define the objects that belong on each respective list, select the pull-down for List One or List Two and you will see the following options:

- **All Objects:** Includes all parts that are currently loaded in your assembly. This is an active list; in other words the system reevaluates the list each time it is run to determine if new parts have been added or removed.

- **All Visible Objects**: Includes all objects that are displayed.

- **All But Selected Objects:** Allows you to specify which parts are not to be analyzed and then add everything else to the list.

- **Class Selection**: gives you the regular class selection dialog to choose the objects to add to the list.

Whether using one list or two lists, you can use the Show option to display exactly what objects are to be included in the respective lists. In addition, any objects designated for one or two lists will appear in their own folder

in the Clearance Browser.

Default Clearance Zone

The default clearance zone field lets you define a value or expression that is to be used as a default clearance area around components/bodies that are to be analyzed. Once a default clearance zone has been specified for a clearance set, there is no way to delete it without deleting the clearance set. The value of the default clearance zone is, by default, set to 0, however the default value can be set in the customer defaults file:
Assemblies_ClrAnlDefClrZone_EU: 0.0 ! inches
Assemblies_ClrAnlDefClrZone_MU: 0.0 ! mm

Analysis Mode

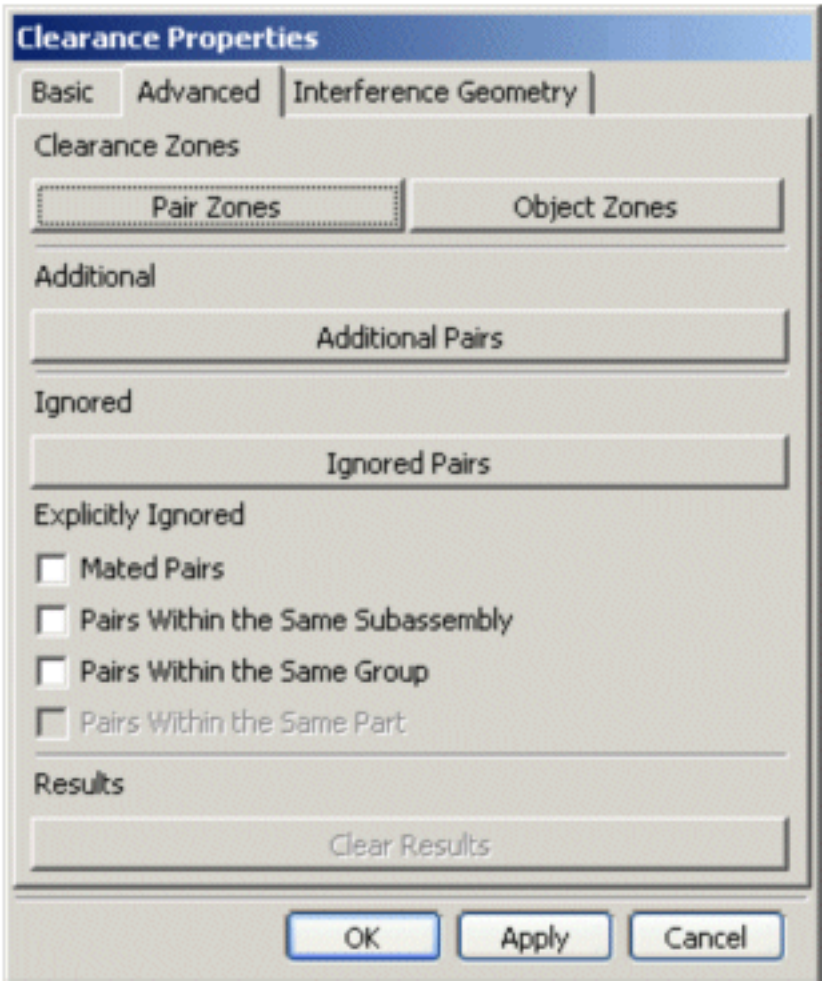
The system gives you a choice of what type of analysis will be executed.

- **Facet Based:** the fastest analysis mode. If there are any unfaceted solids, they will be faceted for the analysis, then deleted. Any detailed analysis of the results should be undertaken by using the Study Interference option. Note: Faceted geometry is created in the Work Part only; if a subassembly is the work part, its component geometry will not be faceted.
  Advantages: Fastest Mode
  Disadvantages: Creates a lot of interference objects. Gives least information back to user. Only as accurate as faceting tolerance.

- **Use Solids Where Available**: (the default) will check loaded solids; any faceted geometry will be checked as in facet based mode. This mode is especially suited for situations where only those objects which require solid accuracy are checked using solids.
  Advantages: Optimum balance of performance / functionality. Facilitates narrowing of analysis to specified solids.
  Disadvantages: Generates facets only in Work Part. Does not check faceted objects created from faces.

- **Use Solids:** is based on the mathematically exact solid geometry within your model. Solid geometry will be loaded for any faceted geometry.
  Advantages: Yields the most exact results.
  Disadvantages: Slowest mode Yields intersection solids and minimum distances.

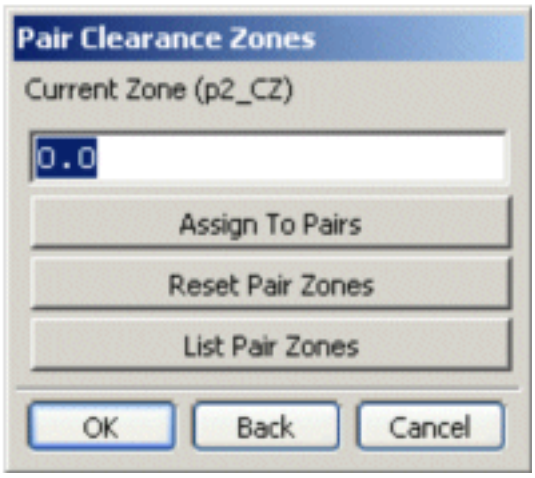
The Advanced Tab



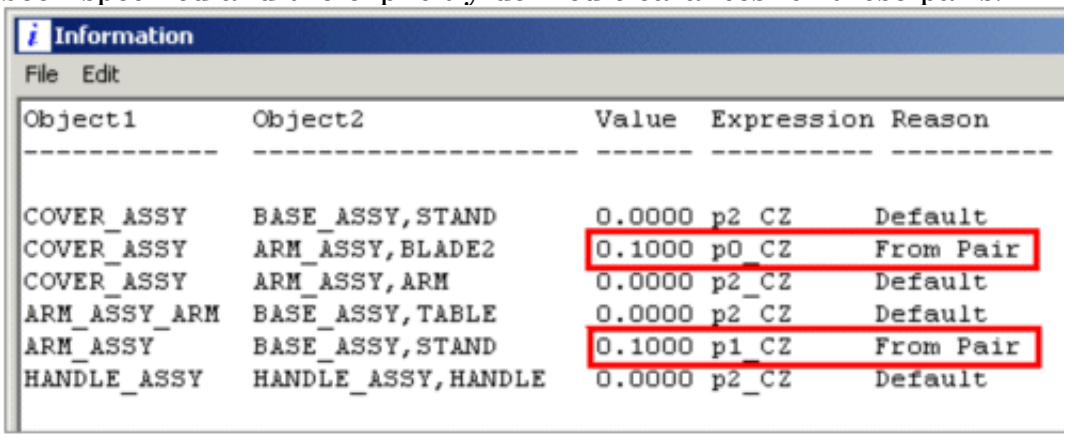
The Advanced page of the Clearance Properties dialog deals with customizing your explicitly defined clearance zones.

Pair Zones

The Pair Zones options brings up the Pair Clearance Zones dialog:



Here you can explicitly designate a clearance zone to object pairs. When the Assign to Pairs, Reset Pair Zones and List Pair Zones options are selected, the Select Objects dialog appears to identify the object pairs affected. List Pair Zones, in conjunction with the Select Objects dialog, will present an Information window delineating whatever pairs have been specified and the explicitly defined clearances for those pairs.



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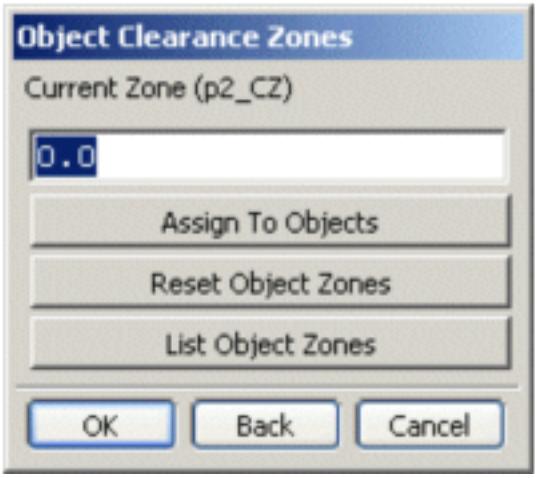
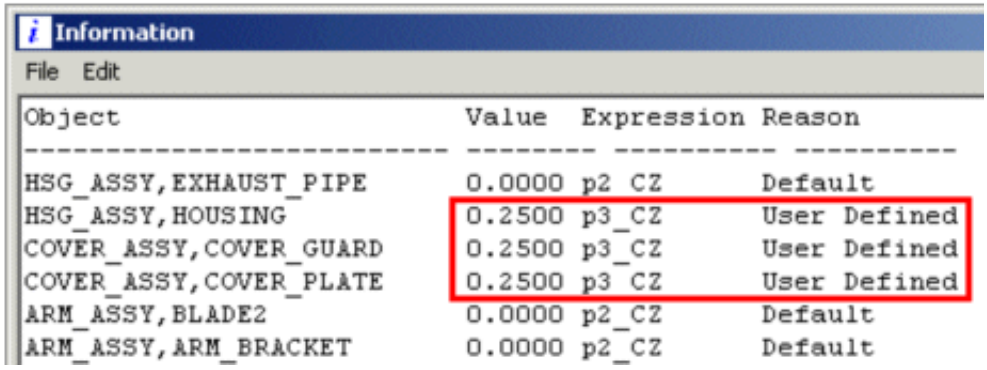
Object Zones

The Object Zones options brings up the Object Clearance Zones dialog:



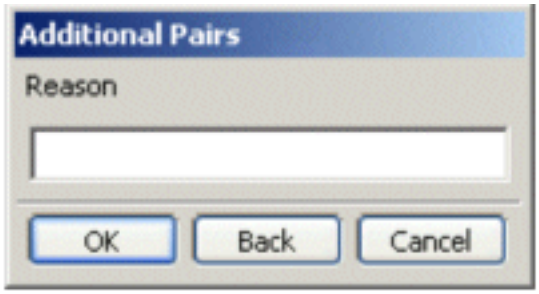
Here you can explicitly designate a clearance zone to individually specified objects. When the Assign to Objects, Reset Object Zones and List Object Zones options are selected, the Select Objects dialog appears to identify the objects affected.
List Object Zones, in conjunction with the Select Objects dialog, will present an Information window delineating whatever objects have been specified and the explicitly defined clearances for those pairs.



An object must reside on your "List" in order for it to be selectable for clearance zone assignment.

Additional Pairs

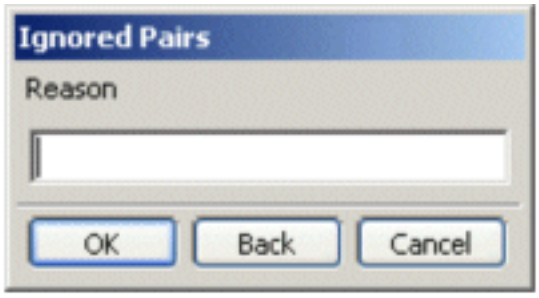
The Additional Pairs option is available if you want to add explicitly included pairs other than those on your lists via the Select Objects dialog. Once specified, you will see the Additional Pairs dialog where you can specify a reason why the designated pairs are being included.



Additional Pairs will be added to the Additional Pairs to Check folder in the Clearance Browser.

Ignored Pairs

The Ignored Pairs options lets you specify object pairs or pairs that are not to be considered for analysis via the Select Objects dialog. Once specified, you will see the Ignored Pairs dialog where you can specify a reason why the designated pairs are being ignored.



Ignored Pairs will be added to the Ignored folder in the Clearance Browser.

Explicitly Ignored

One aspect of ignoring certain pairs is the ability to ignore classes of pairs:

- **Mating Pairs**: automatically excludes pairs of components that are mated together.
- **Pairs Within the Same Subassembly**: allows you to exclude components that come from the same subassembly so they are not checked against each other.
- **Pairs Within the Same Group**: excludes pairs of components that come from the same group.
- **Pairs Within the Same Part**: allows you to exclude bodies that come from the same part so they are not checked against each other . Note that this option is unavailable when doing a component-based analysis.

Clear Results

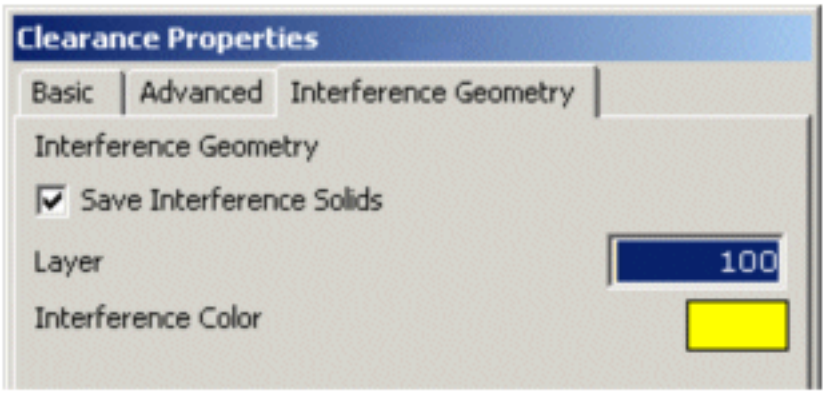
The Clear Results option deletes all the interference results and marks all the objects in the clearance set as out-of-date. This is to allow you to run a complete reevaluation of the current clearance set.

Clearance Zone Hierarchy

Since there are multiple ways the system may assign a clearance zone value, there needs to be a hierarchy of sorts so the system will know what value to use for a given object. The rules are as follows:

- use the zone value assigned to the pair, if not defined then....
- use the larger of the zones of the two objects being considered, if neither has a zone assigned then .....
- use the default zone value.
- List Pair Zones: allows you to display clearance zone information for selected sets of objects.

Interference Geometry Tab



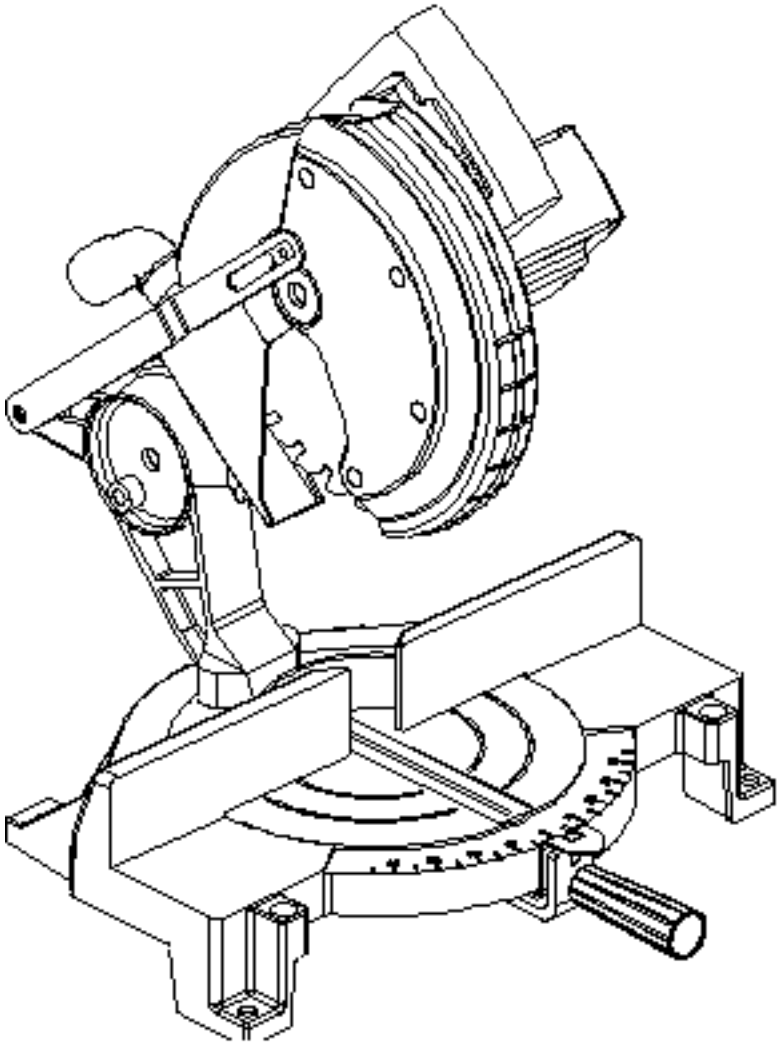
On this page of the Clearance Properties dialog, you specify how you want to view any interference geometry and if you want to save interference solids. All three options here can be controlled using customer defaults.

- Save Interference Solids (determines if interference solids are to be saved.)
- Layer (layer number on which all new interference geometry is placed. A value of zero indicates that the work layer (at the time the clearance set is created) will be used.)
- Interference Color (the color assigned to all new interference geometry)

Activity: Interactive Clearance Analysis

In this activity we will perform a clearance analysis on a simple assembly to demonstrate each type of interference. The student should become familiar with:

- Setting up a Data Set for analysis
- Defining Clearance Zones
- Analysis Modes
- Excluding Objects
- Review Functions



Setting up for the Activity

Step 1: Set your Load Options and open the assembly "saw_cl_anl" from the saw directory.

- ☐ Choose File→Options→Load Options.
- ☐ Be sure the Load Method is set to From Directory.
- ☐ Change the Load Components to All Components.
- ☐ Choose Default Reference Sets.
- ☐ Choose Entire Part.
- ☐ Choose Move Up to place the Entire Part Reference Set to the top of the list.
- ☐ Choose OK.
- ☐ Open the assembly "saw_cl_anl" from the saw directory.

You have learned, during the clearance analysis process, the system will create interference geometry in your work part. Therefore, we are using the Master Model approach and analyzing the application specific part rather than the main assembly.

Step 2: Start the Clearance Browser and create a Clearance Set

- ☐ Select Analysis→Assembly Clearance→Clearance Browser.
- ☐ In the Clearance Browser window, use MB3→Clearance Set→New.

Step 3: Set your Interference Geometry criteria

- ☐ Choose the Interference Geometry tab in the Clearance Properties dialog
- ☐ Set the Save Interference Solids switch to on.
- ☐ Set the Layer that the interference solids will be save on to layer 100
- ☐ Set the Interference Color to White.
- ☐ Select Apply.

These properties are saved within the clearance set, and are persistent for that clearance set unless edited.

Step 4: Set the Clearance Zone for the saw blade.

- ☐ Choose the Advanced tab in the Clearance Properties dialog.
- ☐ Choose the Object Zones option.
- ☐ Enter .25 in the Current Zone entry field.
- ☐ Choose the Assign to Objects option
- ☐ Using the Class Selection menu select the saw blade from the graphics area.
- ☐ Choose OK twice

Since you have set a clearance zone of .25 for the saw blade, you should set up the clearance set not to check the saw blade against the component to which it attaches.

Step 5: Ignore the component pair BLADE2 and UPPER_ARM.

- ☐ Select Ignored Pairs from the Clearance Properties dialog.
- ☐ Using the Select Objects dialog, select the saw blade and the upper_arm (magenta component).
- ☐ Choose OK.
- ☐ Type in the reason for exclusion of the pair e.g. "blade attachment".
- ☐ OK the Select Objects dialog
- ☐ Choose Apply on the Clearance Properties dialog.

Next, we will execute our analysis, since this assembly contains both solids and facets, we can use the Facet/Solid analysis mode.

Step 6: Set up for your clearance analysis.

- ☐ Choose the Basic tab in the Clearance Properties dialog.
- ☐ Key in saw_analysis_1 for the Clearance Set name.
- ☐ Make sure Clearance Set Type is set to Component.
- ☐ Make sure Lists to Check is set to One List.
- ☐ Make sure Analysis Mode is set to Use Solids Where Available.

☐ Choose OK.

Step 7: Start the analysis run

☐ In the Clearance Browser, place the cursor over the SAW_ANALYSIS_1 clearance set, then use MB3→Perform Analysis.

Once the analysis is run, the results are available in the Clearance Browser.

Step 8: Sort the analysis data by Type

☐ Click on the Type column in the Clearance Browser.

Note the soft interference and the distance of encroachment into the .25 clearance. Also note the components in the Ignored folder and the corresponding text. In addition, you can review summary information using the Clearance Browser.

5

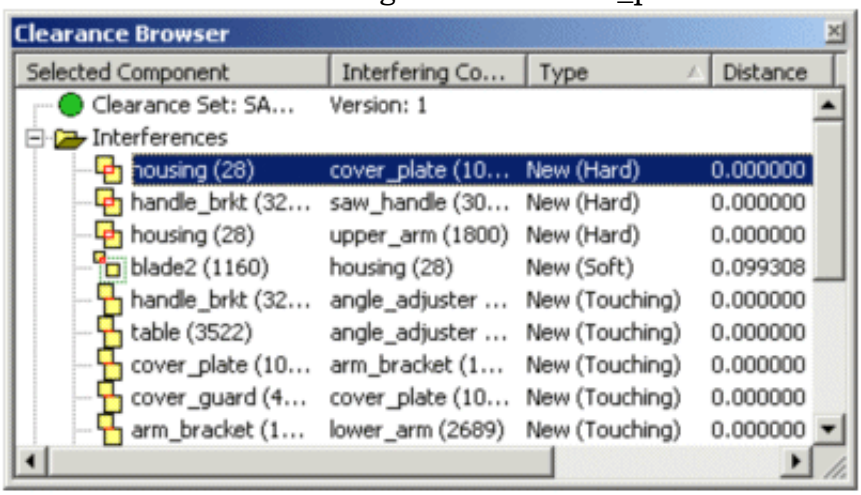
Step 9: Review the summary report of analysis data

☐ In the Clearance Browser, place the cursor over the Clearance Set node, then use MB3→Report.

☐ The results contain a Summary of the analysis (pairs checked, type of analysis etc.) and a more detailed report delineating where the interferences occurred.
The same information can be attained outside of the Clearance Browser by using Analysis→Assembly Clearance→Analysis→Report.
You can also save this report as a text file.
Dismiss the Information window.

Step 10: Examine the interferences using the Clearance Browser.

☐ In the Clearance Browser, select the hard interference node between the housing and the cover_plate.



☐ Use MB3→Study Interference

The housing is shaded and the cover_plate is displayed in highlighted wireframe with the interference in white. Note the right-pointing arrow in the Clearance Browser signaling the node under examination.

☐ With the cursor still over the selected node, use MB3→Flip Shading

Note your other visualization options using MB3: Blank Left/Right, Wireframe Left/Right, Reset Shading

☐ With the cursor still over the selected node, use MB3→Restore Component Visibility.

Step 11: Experiment using these options with other interference nodes within the Clearance Browser.

Step 12: Examine the interference geometry on Layer 100

☐ Using the Layer Settings dialog, make Layer 100 the Work layer and make Layer 1 Invisible.

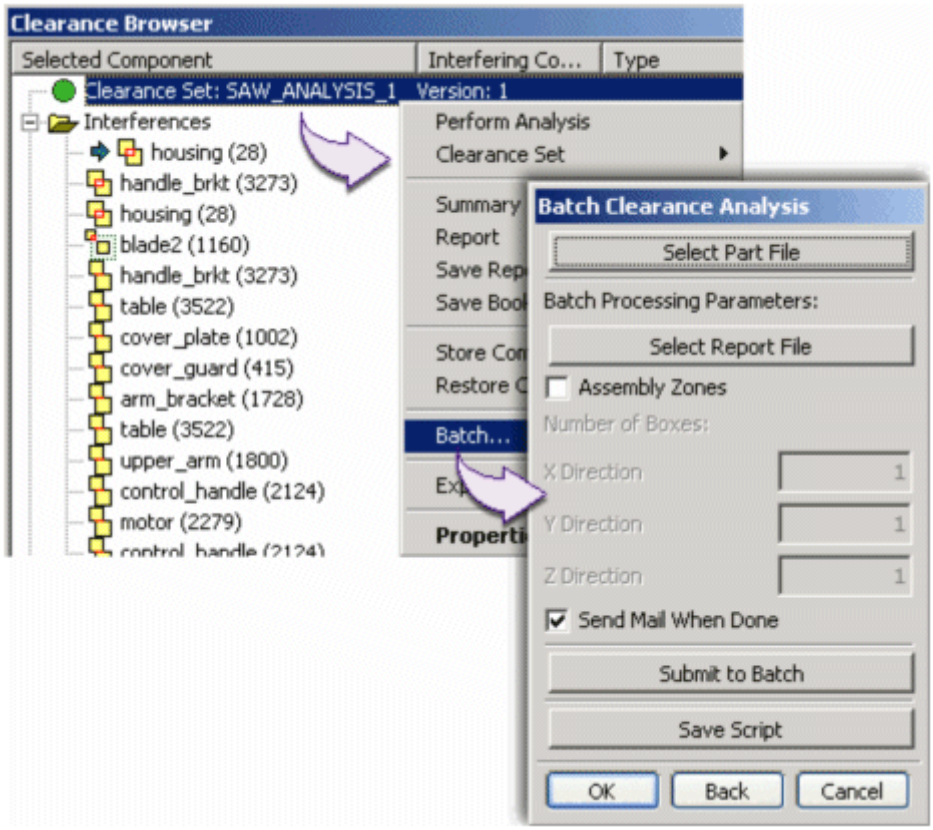
☐ Examine the interference solids and points. (Remember, Touching interferences will show as points and Hard interferences will be solids.

☐ Close all parts.

Batch Clearance Analysis

As you have experienced, a clearance analysis can occupy some time when executed interactively. When working with larger assemblies it is probably preferable to have the analysis done in the background so your productivity is uninterrupted.

The Batch Clearance Analysis dialog can be accessed by using MB3 while the cursor is over the Clearance Set node in the Clearance Browser.



Batch Clearance Analysis locates the components using the saved Load Options.
During a batch submission, the system will internally open the designated assembly and analyze the selected data set using the parameters and preferences stored within it. The results of the analysis are also saved within the part file.

From the main Clearance Analysis dialog, you may select Batch to begin a batch submission.

Batch Submission

Batch submission Options:

- Select Part File allows you to choose what part you want analyzed. The part need not be loaded interactively.

- Select Report File allows you to designate the file name that the clearance analysis textual data will be stored in.

- Assembly Zones: This option allows you to specify that an assembly be divided up into zones and the zones analyzed one at a time. You would input the number of "box" zones in the X, Y, and Z directions. The system would load only the components in the first zone and analyze them, it would then unload those components and load the components for the next zone and so on. This option is recommended to be used only when your assembly is so large that it can not be loaded all at the same time. If performed on a smaller assembly, this option actually lengthens the process.

- Send Mail When Done tells the system to send you operating system mail upon the completion of the batch program. The content of a typical mail message would include the following:
  Batch Clearance Analysis completed..... successful
  Part: ...../your.prt
  Data set: ALL Report File: .....report.log

- Submit to Batch starts the batch process.

  Permission for batch submission is controlled by the files /usr/lib/cron/at.allow and /usr/lib/cron/at.deny. If your user name is not properly configured in these files, your batch command will not be submitted and an error will be displayed. Your system administrator can make those changes for you.

If you find that you do not have batch submission privileges as described in the above note, there are two alternate methods of submitting a batch job:

- Alternate Option 1: Creating, then submitting a batch script from a terminal window.

  Save Script creates a script which can be executed from a terminal window. The system will prompt you to name the script and designate what directory it is to be created in.

- Alternate Option 2: Command line submission done from a terminal window.

  This option is discussed more fully in Appendix B.

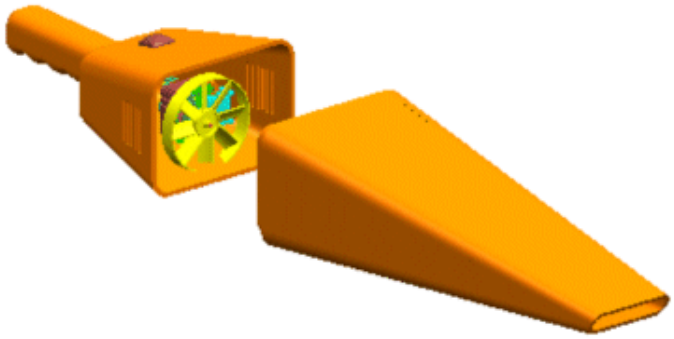
Both of the alternate options require the environment be configured properly. One criteria the batch clearance analysis program depends on is that you have the "ug libraries" in your PATH.

There is an alternate way of setting your environment.
The following command executed from a terminal
window will set your environment properly:
$ export $ export SHLIB_PATH=/usr/lib:$ugii_base_dir/ugii/uglibs
or depending on your shell type;
$ setenv SHLIB_PATH /usr/lib:$ugii_base_dir/ugii/uglibs

If you find yourself using this method often, you should consider creating a script that sets your shared library path for you

Activity: Batch Clearance Analysis

In this activity, you will concentrate on the steps to initiate a clearance analysis in batch mode. The intent is to give you practice working with a batch clearance analysis and how to view and use the results.



During this activity you will simulate the steps you might take if you were reviewing an assembly that you did not create (i.e. you do not have write access to it).

Running a Batch Clearance Analysis

Step 1: Create a new assembly.

- ☐ Create a new part called "*dust_cl_anl*"
- ☐ Set your load options to From Directory.
- ☐ Add the part "dust_vac.prt" from the "vac" directory to the new assembly as a component.

This is the assembly on which you will execute your clearance analysis.

Next set up the preferences and parameters for your data set inside your assembly. If we preferred, we could just accept all of the default parameters and preferences, however; it is probably more useful to designate what we want.

Step 2: Create a Clearance Set

- ☐ Invoke the Clearance Browser: (under the Analysis pull-down menu) then choose Analysis→Assembly Clearance→Clearance Browser
- ☐ In the Clearance Browser, use MB3→Clearance Set→New..

Step 3: Set the Basic properties: Set List 1 so that it contains all objects except the nozzle.

- ☐ Set the Clearance Set Name to batch_test_1.
- ☐ Make sure One List is selected.
- ☐ Choose the All But Selected Objects option in the List One pull-down,
- ☐ Choose Edit
- ☐ Pick the Nozzle from the screen, then OK.
- ☐ Choose Apply.
- ☐ Cancel the dialog.

Step 4: Set your Interference Geometry Properties.

- ☐ Using the Interference Geometry tab, set your interference layer to 15.
- ☐ Set your Interference Color to White.
- ☐ Set Save Interference Solids to on.
- ☐ Choose Apply

Step 5: Save your assembly

In order to access the batch submission menu for clearance analysis, a part must be loaded. Remember the system actually loads the assembly internally. It will use your saved load options found in the file load_options.def. If the assembly is unable to load the part file using those load options, you will get a failed clearance analysis.

Step 6: Create a temporary part called anything you like.

- ☐ With the cursor over the Clearance Set node, use MB3→Batch.
- ☐ Choose Select Part File
- ☐ Choose your dust_cl_anl part file, then OK.

If you do not use the option Select Report File, and specify a file name the system will name the new file "report.log" by default.

Step 7: Choose Submit to Batch

- ☐ Confirm any confirmation messages you will get.

Your batch process has now been submitted.

Step 8: Note: If your Batch Submission was not successful execute the following steps:

☐ Select Save Script and enter the name "clranl.cmd".

☐ Create a terminal window using the method described on page .

☐ Execute the Script by typing at the prompt: $ ./clranl.cmd for a UNIX machine (type $.\clranl.cmd for an NT machine).

When your clearance analysis finishes, you will normally receive operating system mail notifying you of its completion. The message tells you the status of the analysis, whether it was successful or not, the part file checked, the name of the data set checked, and the location of the report file.

Reviewing a Batch Clearance Analysis

You can review the result of a Batch Analysis from the operating system by inspecting the file "report.log". This can be done using the text editor of your choice or by the OS command: $ more report.log.

You can also retrieve the part that was analyzed and inspect the interferences, report, etc. within the part file.

Step 1: Open your clearance analysis file and review the results. How many Hard interferences do you find?

A batch analysis can also be initiated from the command line of the OS. The technical documentation for this process has been included in the appendix.

Summary

This lesson taught you how to determine the interferences of all components in one operation.

In this lesson you:

Used various modes of interference.

Used various analysis modes.

Performed a Clearance Analysis and Interpreted the results.

Executed a Batch Clearance Analysis.

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Lesson

6 *Assembly Weight Management*

Purpose

In this lesson you will learn how to use Assembly Weight Management to optimize the process of monitoring the mass properties of large assemblies.

Objectives

Upon completion of this lesson, you will be able to:

Generate caches of analysis information on piece parts.

Assert the mass of a component with a known weight.

Set a "Not to Exceed" weight limit on an assembly.

Define a reference set for calculations

6

Assembly Weight Management

Using Assembly Weight Management provides an alternative to the existing functionality found in the Analysis→ Mass Using... options. You will learn how to calculate the weight of components that are not loaded and how to define the properties of parts that may not be accurately modeled (like purchased parts).

Benefits of Assembly Weight Management

NX Assembly Weight Management provides support for users wishing to evaluate weight and other mass properties in large assemblies.

By using Assembly Weight Management in the intended manner, customers are able to quickly calculate the mass properties of assemblies even if :

- some of the components are not loaded
- the assembly is large and complex
- some of the components have published mass properties but no accurate solid model (eg. purchased parts)

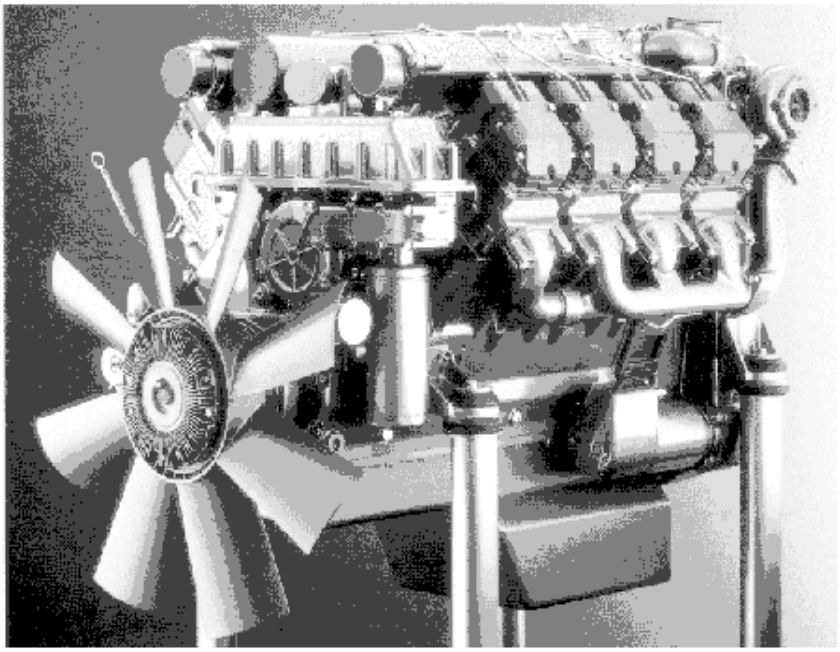
  It is also possible to set weight limits on individual parts and to be alerted if the limits are exceeded.

When to Use Assembly Weight Management

Assembly Weight Management is part of the Advanced Assemblies module. It complements the existing Analysis→Mass Using... options. It is not a replacement for it.

The Analysis→Mass Using...options do not provide the performance and flexibility benefits of Assembly Weight Management. It is suitable for casual use in smaller assemblies. It requires the solids to be loaded and visible.

This contrasts with Assembly Weight Management, which uses other methods to specify which solids are included in the calculation and does not usually require all of them to be loaded or visible.



The performance of Assembly Weight Management calculations is greatly improved when NX is able to re-use the results of previous calculations (Caching of Results is discussed later). This is possible for components where the following three conditions apply:

NX has previously calculated and stored the weight of a component.

The component has not changed since the calculation was made.

The previous calculation was performed at an accuracy at least as high as the new calculation.

Default Accuracy

Although the value of accuracy can be controlled interactively in the Assembly Weight Management menu, the initial value is set by the customer default Assemblies_WeightDataAccuracy. It is therefore recommended that a suitable default value is entered in the customer defaults file and used whenever possible during weight calculations.

Optimizing Performance

Caching of Results

NX Assembly Weight Management enables the fast generation of assembly mass property information by storing and re-using the results of previous mass property calculations. This technique is known as "caching".

By making good use of caching, it is possible to quickly generate mass properties for even the largest of NX assemblies. To make the most of caching, it helps to understand how caches are created and maintained, and when they are used.

Generating Caches

For simplicity, assume for a moment that only piece parts contain solid bodies and that assemblies reference components without having any solid bodies of their own. If your work part is a piece part and you calculate its mass properties using Assembly Weight Management, a cache will be created for that piece part. Piece part caches are saved with the piece part. If the Generate Weight Data save option is used, NX will ensure that an up to date cache is saved with the piece part (discussed later in this chapter).

Piece part caches are available for re-use in the subsequent calculation of mass properties for that piece part or any assembly referencing that piece part as a component.

If your work part is an assembly, an Assembly Weight Management calculation will create a cache for all of the components involved in the calculation. These caches are usually stored with the work part, not with the piece parts.

These component object caches are only available for use in subsequent mass property calculations for the same assembly. They cannot be accessed by components of that assembly, higher level assemblies, or by the piece part itself.

If Assembly Weight Management is used for an assembly, NX will use any available component or piece part caches which are accurate enough, in order to speed up the mass property calculation. Update Structure should be used prior to the calculation to ensure that caches for unloaded components are all available and up to date.

Invalidating Caches

NX generally knows when a Assembly Weight Management cache is no longer valid (for example when the solid geometry is modified). When this happens, NX deletes the cache to prevent it from being used.

An Assembly Weight Management cache will become invalid if its related component is modified. Therefore, run Update Structure before calculating the assembly mass properties to ensure all caches are current.

Asserting Properties

When you use the option to assert the mass properties of a part, you have the option to assert some of the properties as "unknown". For example you can specify the weight of a part, but declare that the volume and density are not known. However if you do this, the same properties will be reported as "unknown" for any assembly which contains that part. In order to prevent too many cases where some assembly mass properties are unknown, it may be best to insist that certain weight properties are specified for all parts with asserted mass properties. This would need to be set up as a company standard practice. As this activity continues, you will witness the behavior of Assembly Weight Management when some of the values are asserted as "unknown".

Activity: Assembly Weight Management

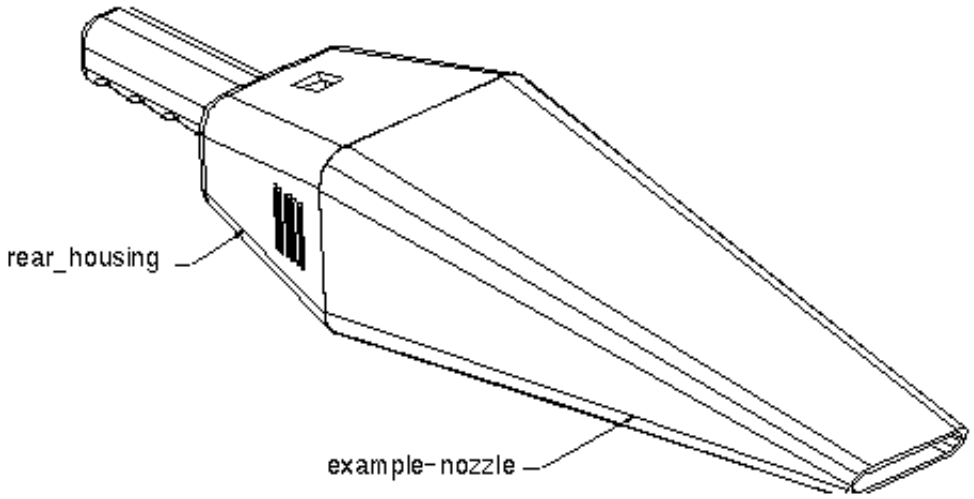
In this activity, you will assert the mass of a purchased component, set a weight limit and calculate the weight of the vacuum. Then you will make a modeling change and check the weight again.

Step 1: Open the assembly "dust_vac" from the "vac" directory and load the BODY reference sets of all of the components.

- ☐ Select File→Options→Load Options→Default Reference Sets
- ☐ Type in "BODY" and press return
- ☐ Choose OK
- ☐ Select File→Open
- ☐ Choose the "dust vac" partfile from the "vac" directory.

Loading the BODY reference set has no effect on what is used for an Assembly Weight Management calculation. The reference set used is determined by setting the Define Reference Set button in the Assembly Weight Management dialog box.

The two large components of this assembly (housing and nozzle) and the switch button have had their density defined as a plastic



Step 2: Verify the density of either the nozzle or the rear_housing

☐ Choose Information→ Object and select object (check the que line to be sure you pick the solid body)

☐ Choose OK.

The density should read : Density (G/Cu Millimeter) =0.00140

☐ Dismiss the Information window.

The system caches the mass data by assigning a system attribute $mass.

The $mass attribute cannot be listed in any report.

With all of the components loaded, you can calculate the weight of the assembly. However, this creates a $MASS cache on the component objects which will not be available in any other assembly.

It is more desirable to have the $MASS cached on the piece parts so that the $MASS is accessible for other assemblies using these parts.

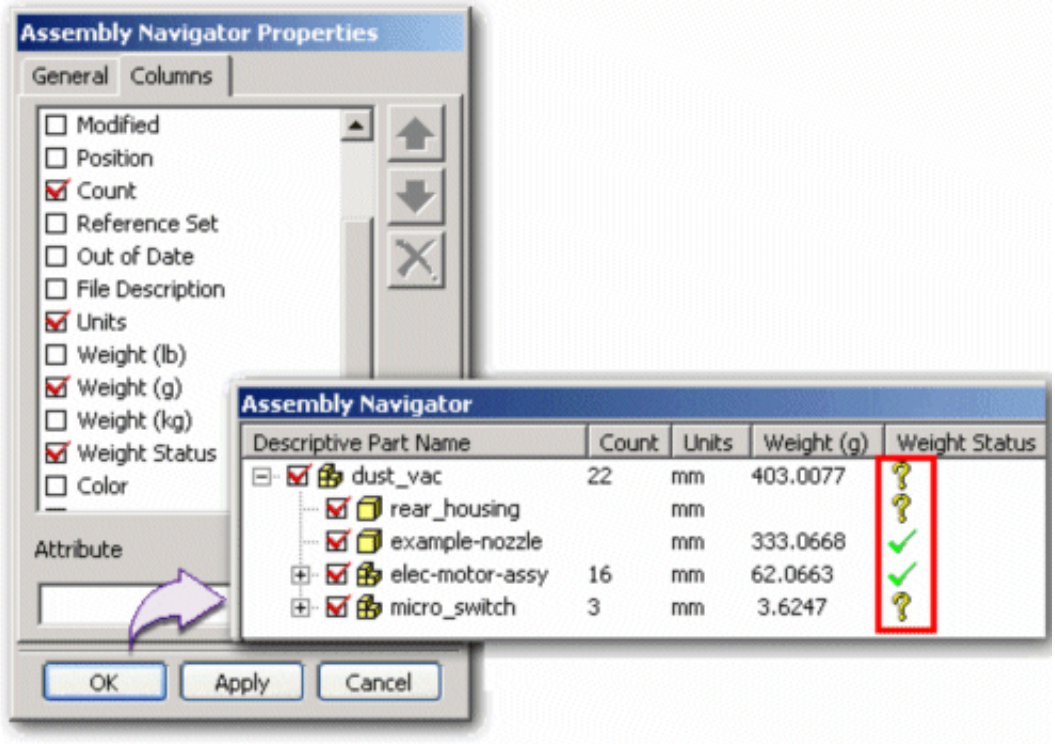
Step 3: Open the Assembly Navigator and check Weight status

☐ Choose the Assembly Navigator tab. (Unix users, choose the Assembly Navigator icon on the Navigators toolbar.)

☐ With the cursor over the column descriptor banner, use MB3→Properties to invoke the Assembly Navigator Properties dialog.

☐ Choose the Columns tab on the Assembly Navigator Properties dialog.

☐ Turn on (check) the Count, Units, Weight (g) and Weight Status columns, then OK.



Note that the current weight status of the rear_housing is deemed to be "unreliable". The weight column information will be useful throughout this lesson.

6

Bottom-up Approach to Assembly Weight Management

To make the most of the caching capabilities of Assembly Weight Management, a "bottom-up" approach to Assembly Weight Management is recommended. By ensuring that all parts maintain caches, pre-calculated mass property data is always available for Assembly Weight Management calculations. The only components that require a live calculation from the solid data are those which have been modified in that session.

To implement a bottom-up approach, you need to:

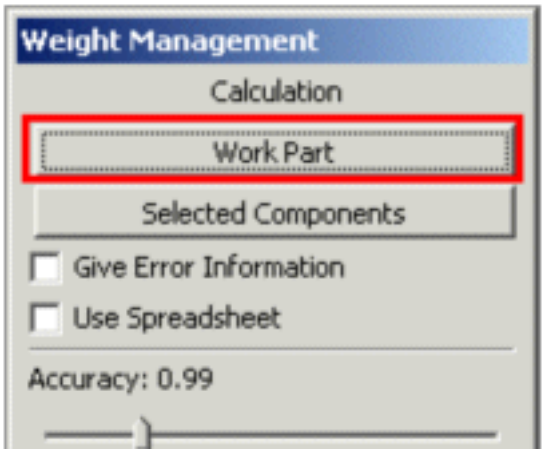
- ensure that the weight reference set is correctly set and maintained for each part
- ensure that the Generate Weight Data save option is used for all parts
- agree on a standard accuracy, so that caches are accurate enough to be used

Step 1: Create a cached $MASS in the "rear_housing", "example-nozzle" and the "button" components.

☐ Make the "rear_housing" the work part

☐ ChooseAnalysis→Assembly Weight Management

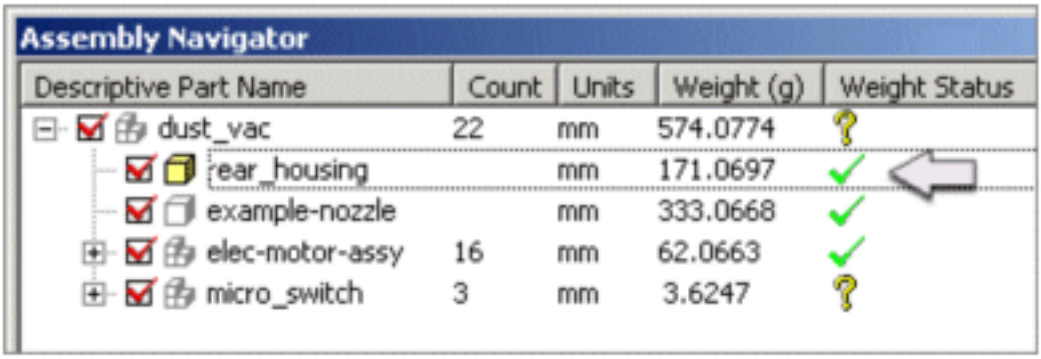
☐ Under the "Calculation" section, choose Work Part.



Note the mass properties delineated.

☐ Dismiss the information window.

☐ Note in the Assembly Navigator that the rear_housing now has a reliable value.



If multiple solids exist in a piece part and you do not want the additional solids to contribute to the weight, you need to set the weight reference set definition in that piece part. By defining which reference set contains the appropriate solid body, the system will create the cache based on the one solid rather than all of the solids.

Step 2: Repeat the process for the " example-nozzle" and the "button" (found under the micro_switch component).

Step 3: Make the dust_vac the work part.

Step 4: Save

6

Asserting Properties

The only component that does not have a $mass at this time is the micro switch. Since it is a purchased part, you will assert a mass to it based on the suppliers catalog.

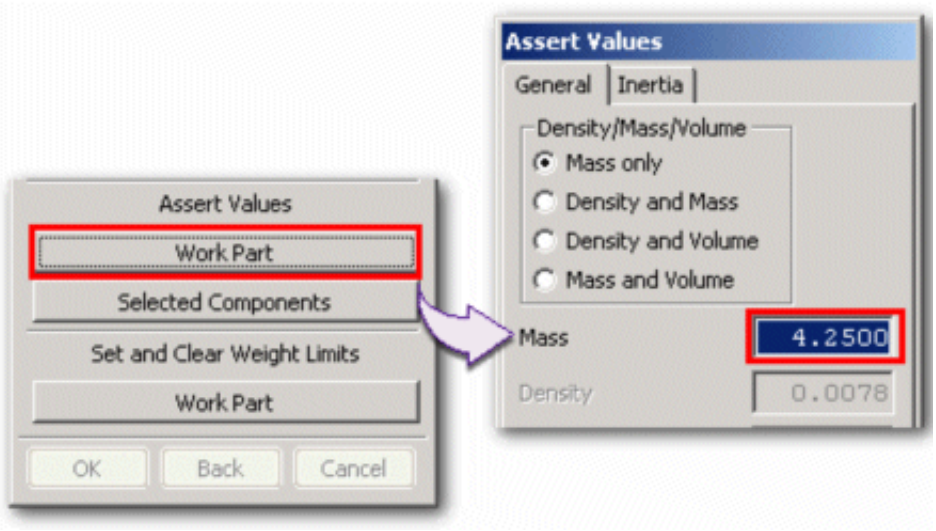
Step 1: Assert a mass to the purchased micro switch named "switch"

☐ Make the "switch" the work part.

☐ Choose Analysis→Assembly Weight Management

☐ Under the "Assert Values" text, choose Work Part

☐ Set the Mass to 4.25 grams



☐ Choose OK

Note in the Assembly Navigator how asserted values are symbolized.

☐ Select the Save icon.

☐ Make the dust_vac the Work Part

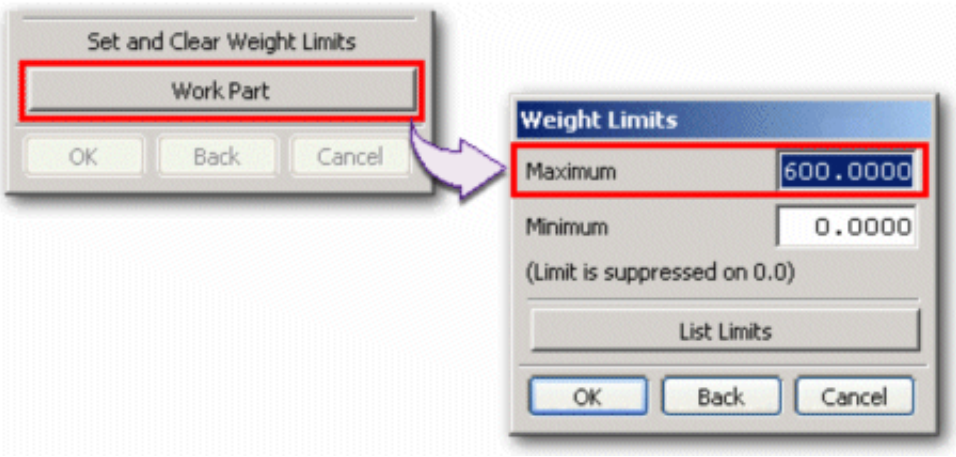
There is a weight that this assembly cannot exceed. You need to set the maximum weight limit for the assembly.

Step 2: Set the maximum weight of the assembly to 600 grams . If the weight exceeds 600 grams, you will be notified in the report.

☐ Choose Analysis→Assembly Weight Management

☐ Under the "Set and Clear Weight Limits" text, choose Work Part.

☐ Enter 600 as the maximum weight limit



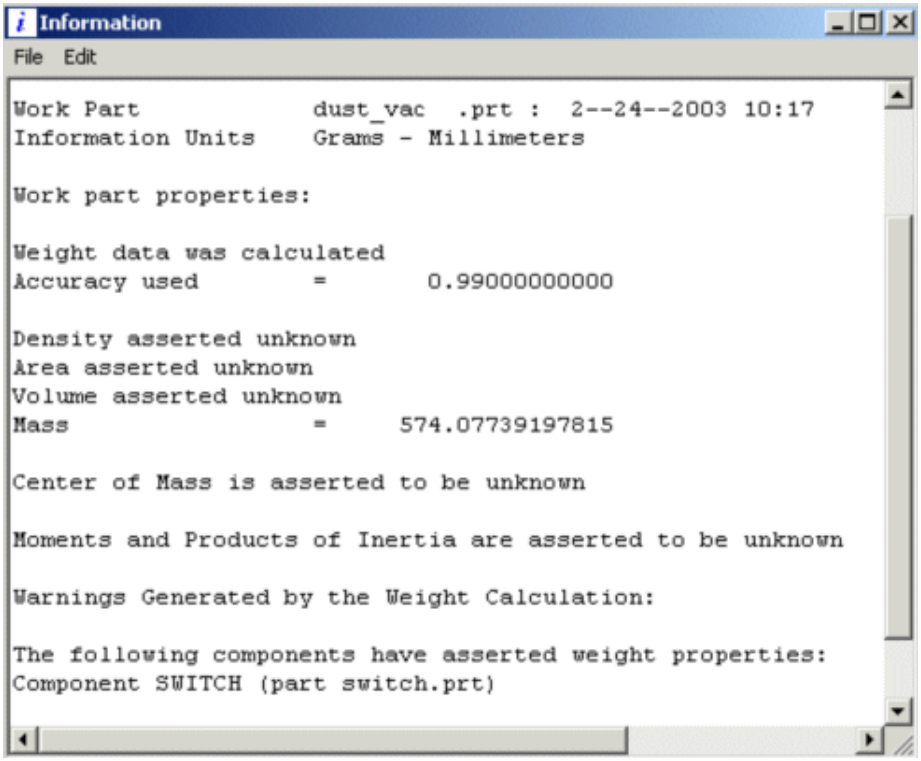
☐ Choose OK

With all of the preliminary set up completed you are ready to calculate the weight of the assembly.

Step 3: Calculate the weight of the assembly using the work part (dust_vac)

☐ Choose Analysis→Assembly Weight Management

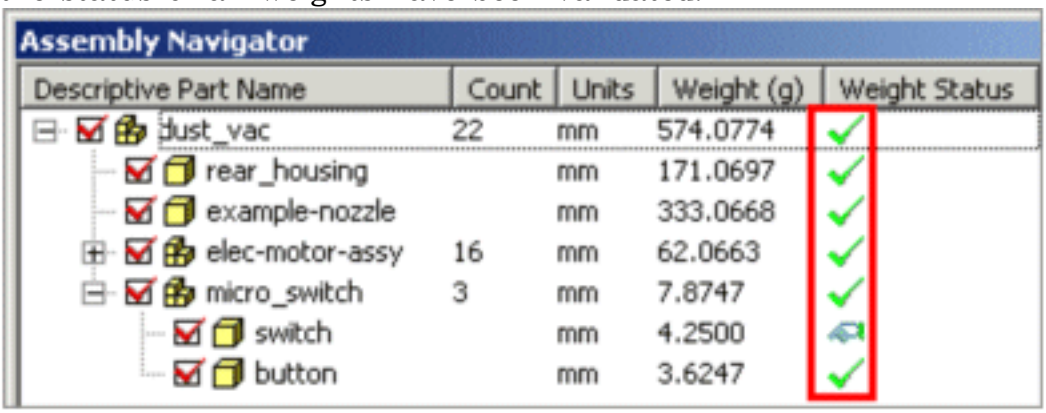
☐ Under the "Calculation" text, choose Work Part.



6

☐ Dismiss the Information window.

Note again in the Assembly Navigator that the status of all weights have been validated.



Because the mass of the micro-switch was asserted but the other properties of the switch were set to unknown, the only usable value is the mass.

Now you will make a modeling change to the rear_housing and the nozzle to make them thicker. Then you will check the weight again.

Step 4: Change the wall thickness of the rear_housing and the "example-nozzle" to 3.5 mm.

☐ Choose the Make Work Part icon from the Assemblies toolbar or choose Assemblies→Context Control→Set Work Part.

☐ Select either the rear_housing or the example-nozzle and choose OK.

☐ Choose the Modeling icon from the Applications toolbar or choose Application→Modeling (If necessary).

☐ Select the Edit Feature Parameters icon.

☐ Double-click on the Hollow feature.

☐ In the Hollow dialog, key in a Default Thickness of 3.5.

☐ Choose OK (3 times).

☐ Repeat this process for the remaining component.

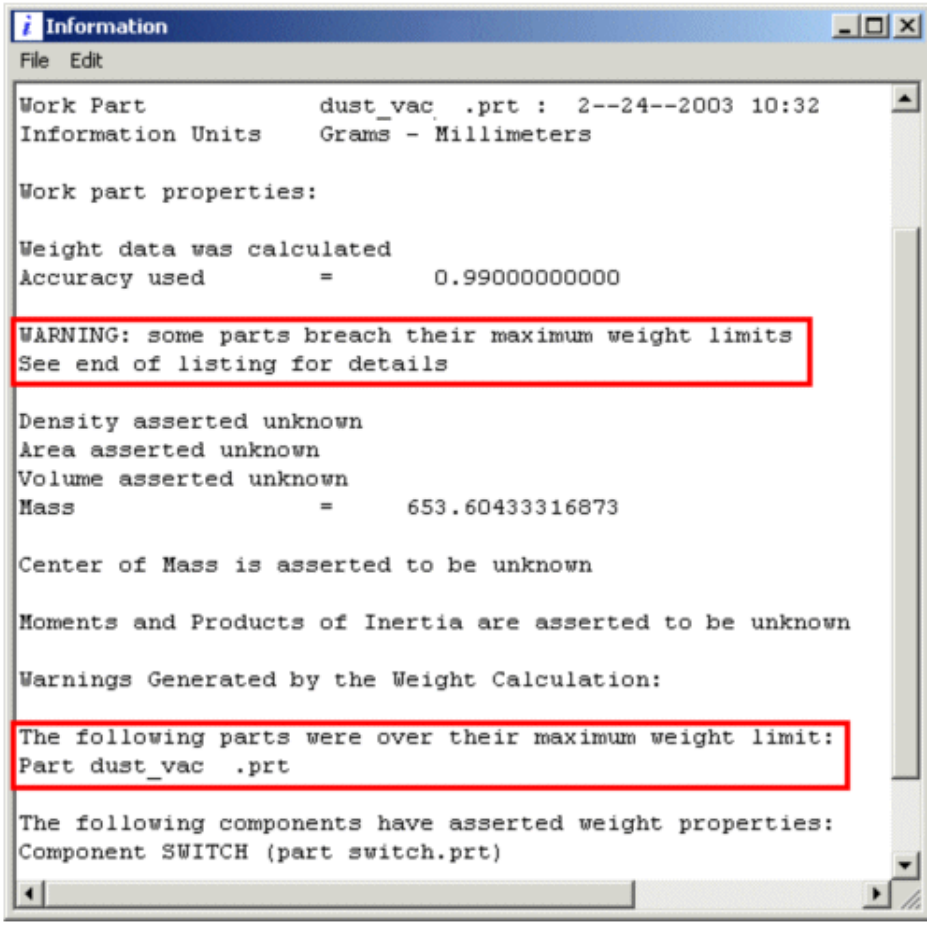
Step 5: Calculate the weight of the assembly using the work part again.

☐ Select Assemblies→Context Control →Set Work Part

☐ Choose the Displayed Part option.

☐ Choose Analysis→ Assembly Weight Management.

☐ Under the Calculation text, choose Work Part.



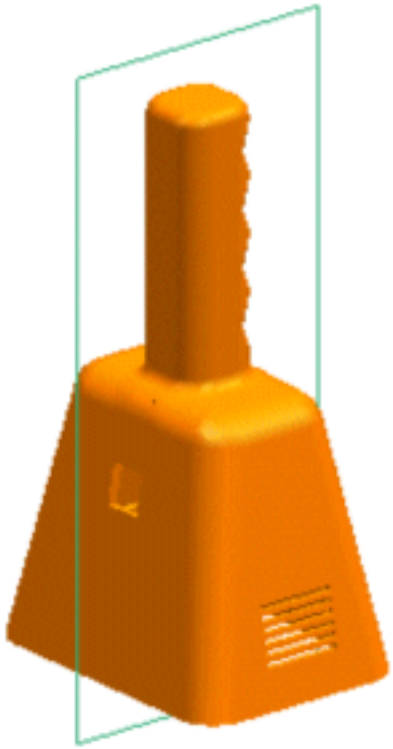
☐ Save and close all parts.

6

Activity: Updating Weight Calculations

Notice that a warning was generated because the changes now cause the part to exceed the maximum weight that was defined for the assembly. Unigraphics NX did not need to recalculate the mass for many components as most had up-to-date caches. In fact, it would have been possible to calculate the weight of the assembly even if many of the components were not even open. In this section, you will open the "rear_housing" and "example-nozzle" and modify their hollow back to the original thickness without having the assembly open. You will then open the assembly without loading any components and recognize the behavior of the cached $MASS value on each component.

Step 1: Open the rear_housing partfile and change the hollow value back to 3 mm.



You will now update the $MASS cache to recognize your change. There are two ways to accomplish the task.

☐ By calculating the weight of the part using the Assembly Weight Management dialog and then save the part.

☐ Or by selecting File→Options→Save Options and toggle the Generate Weight Data option ON

Earlier in this activity you created the cache by asking for the weight calculation and then saving the part. Now you will update the cache by the second method.

Updating Weight Calculations on Save

The weight properties of a part can automatically be updated and stored with the part each time the part is saved with modified geometry. This is achieved by switching on the part's Generate Weight Data save option. Although the results of the weight calculation are not presented to the user during the save operation, they become available for re-use in subsequent Assembly Weight Management calculations involving that part.

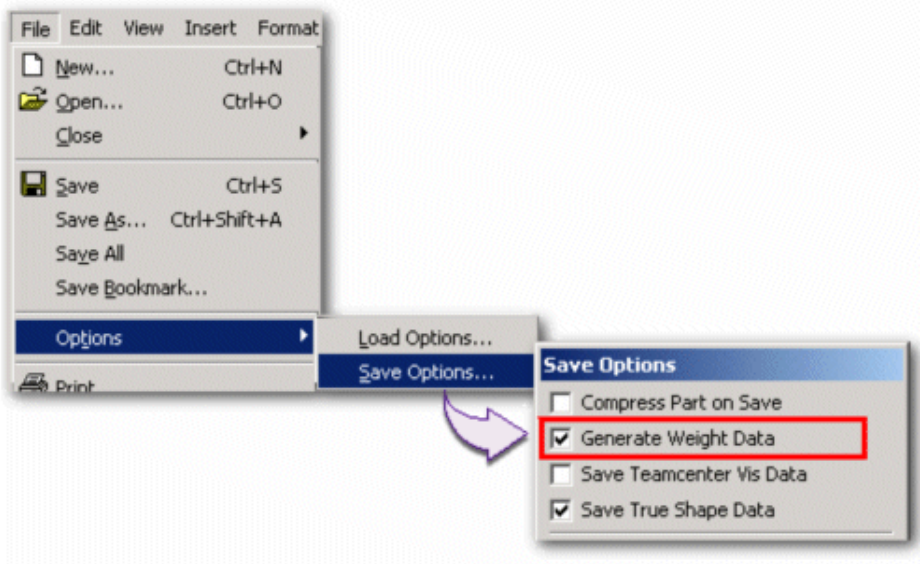
Therefore if a majority of parts use this save option, Assembly Weight Management calculations for an assembly can become much faster as up-to-date weight data is available for most or all of its components. For this reason it may be best to use this save option by default.

By setting Assemblies_GenerateWeightData to yes in the customer defaults file, all newly-created parts and parts which were last saved in a pre-v13 release of UG will initially have the Generate Weight Data save option set. In cases where the weight re-calculation has a noticeable affect on the performance of saving a part, the save option can be turned off for that part and may be restored when the design has stabilized and is saved less often.

Step 1: Enable the Generate Weight Data during a save

☐ Choose File→ Options→Save Options

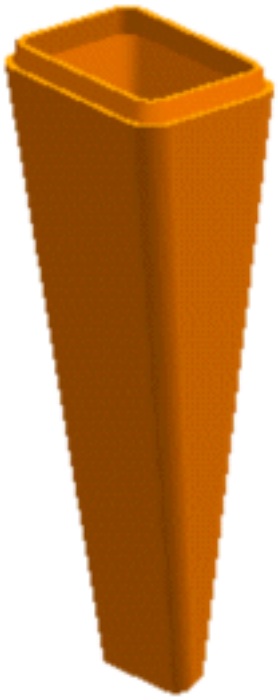
☐ Turn the Generate Weight Data option ON and choose OK.



Step 2: Save the rear_housing

Step 3: Open the example-nozzle

Step 4: Change the hollow value back to 3 mm.



The Generate Weight Data toggle is part file dependent. It must be turned on in each file that you want to have the cache created in.

Step 5: Choose File→Options→ Save Options.

☐ Turn the Generate Weight Data option ON and choose OK.

Step 6: Save the example-nozzle.

Step 7: Close all parts.

Step 8: Set your load options so that no components are loaded when you open an assembly

☐ Choose File→Options →Load Options.

☐ Be sure your Load Components are set to No Components.

☐ Choose OK.

Step 9: Open the dust_vac assembly

Step 10: Check the weight of the assembly without having its components open

☐ Analysis→ Assembly Weight Management

☐ Under the "Calculation" text, choose Work Part

You may receive an Update Out-of-Date Assembly Structure message that asks you if you want to update the cached component weight properties before calculation

☐ Choose Yes.

Notice that the caches have updated and the report is showing the original weight of the components before the wall thickness was changed without having the solids loaded.

Step 11: Dismiss the Information window.

When Multiple Solids Exist in a Part File

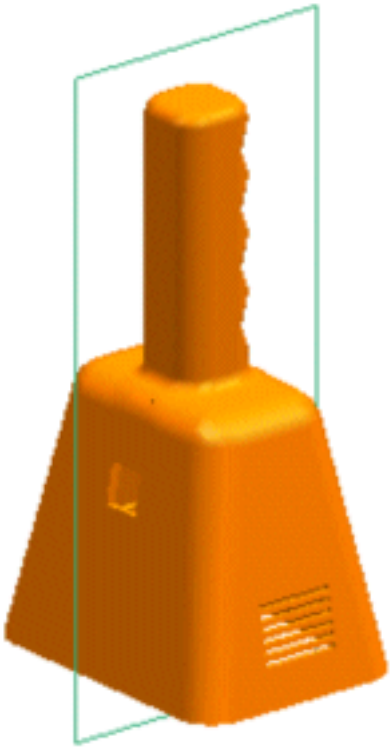
In the next portion of the activity, you will create an associative extracted solid in a component part and experience the effects of the Assembly Weight Management function.

Step 1: Open all of the components of the assembly.

☐ Choose the Assembly Navigator tab. (Unix users, choose the Assembly Navigator icon on the Navigators toolbar.)

☐ Press MB3 on the dust_vac node and choose Open→Assembly.

Step 2: Make the rear_housing the Displayed Part using the Assembly Navigator.



Step 3: Using the Extract Body feature, create an associative copy on layer 10

☐ Change the Work Layer to 10.

☐ Choose the Modeling icon from the Applications toolbar or choose Application→Modeling (If necessary).

☐ Choose the Extract Geometry icon from the Form Feature toolbar or choose Insert→Form Feature→Extract.

☐ Select the Body icon from the Extract dialog box

☐ Select the solid as the target.

☐ Choose OK.

☐ Set layer 1 to invisible (the original solid).

Step 4: Trim the new solid to the Datum Plane, keeping the back half.

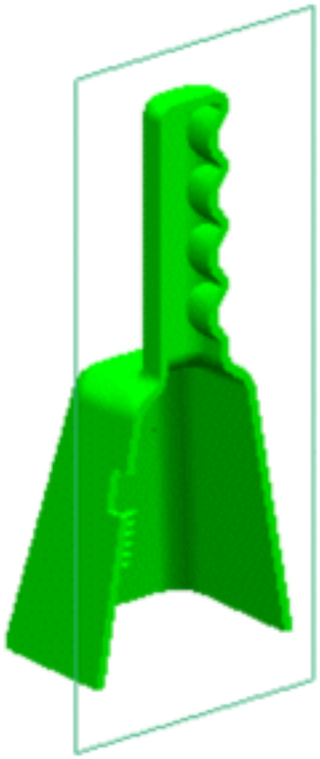
☐ Choose the Trim Body icon from the Feature Operation toolbar or choose Insert→Feature Operation→Trim.

☐ Select the solid as the target.

☐ Choose OK.

☐ Select the Datum plane.

☐ Check the Normal. If it points to the right, accept the default direction, otherwise, reverse the direction.



Step 5: Create a reference set named "pattern" and add the solid

☐ Choose Format→Reference Sets→Create icon

☐ Key in pattern as the new reference set, then OK.

☐ Select the trimmed solid, the OK.

☐ Close the Reference Sets dialog.

☐ Make layer 1 the Work layer and layer 10 invisible

Step 6: Save the rear_housing

There are now two solids in this part file. One is in a reference set named BODY and the other is in a reference set named PATTERN.

Step 7: Make the dust_vac assembly the Displayed Part and the Work Part.

Step 8: Calculate the weight of the assembly again

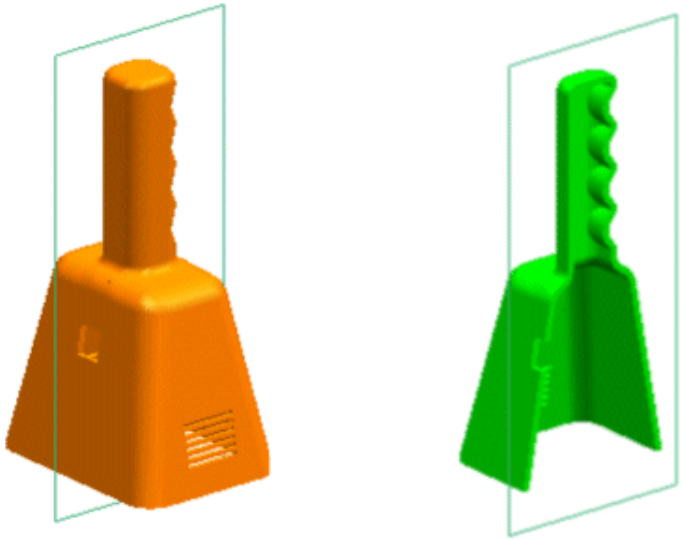
☐ Choose Analysis→Assembly Weight Management.

☐ Under the Calculation section, choose the Work Part button

☐ Examine the weight information.

Notice that even though you are viewing the BODY reference sets for all of the components, the new extracted solid is being added into the weight.

Loading the BODY reference set has no effect on what is used for a Assembly Weight Management calculation. The reference set used is selected under the Set Reference Set button in the Assembly Weight Management dialog.



BODY reference set **PATTERN** reference set

Standard Weight Reference Set

By default, all non-sheet solids in a part contribute to its weight properties (including invisible solids and solids excluded by reference set). In some cases this is not appropriate as parts can contain solids which do not represent the actual part (eg. reference geometry).

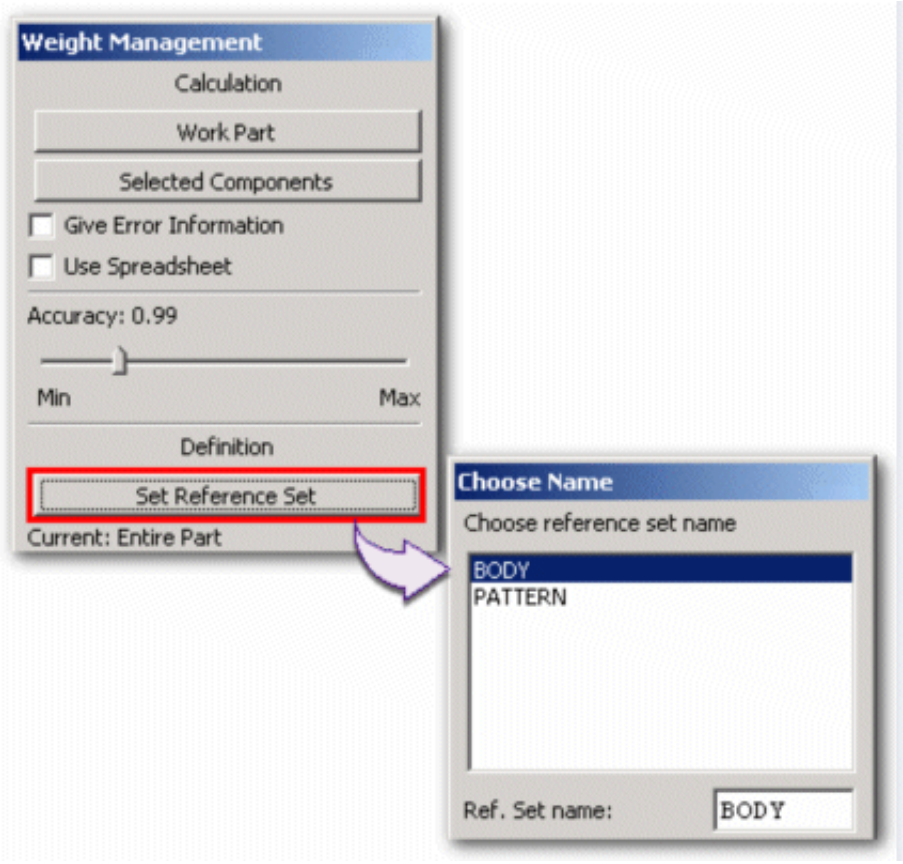
For this reason, in each part it is possible to specify a reference set to use in weight calculations. When that part is involved in a weight calculation, only the solids included in the specified reference set will contribute to the result. The weight reference set must be properly set up for any part that contains "extra solids", if there is any chance that the part will be involved in an Assembly Weight Management calculation.

Therefore, unless it can be guaranteed that a vast majority of parts only contain solids that represent the real part, it is recommended a standard weight reference set is used for each part. This reference set should have a standard name and should contain only those solids that are to be used in weight calculations. Some customers may be able to use an existing standard reference set that includes the same data.

Step 1: Define the reference set to be used by Assembly Weight Management in the rear_housing part.

- ☐ Change the Work Part to the rear_housing.
- ☐ Choose Analysis→Assembly Weight Management.
- ☐ Choose the Set Reference Set button.
- ☐ Choose BODY.

☐ Choose OK.



☐ Save the rear_housing.

☐ Make the dust_vac the Work Part.

Step 2: Calculate the weight of the assembly again.

☐ Choose Analysis→Assembly Weight Management.

☐ Under the Calculation section, choose the Work Part button.

☐ Examine the weight information.

The weight is once again correct in that it shows only the desired components are being used to evaluate the weight.

Common Misunderstandings and Mistakes

Since Assembly Weight Management calculation results are expressed mainly as numbers and have very little graphical representation, it is not always obvious when you have made a mistake setting up the calculation. This section describes some cases where misunderstandings can lead to the generation of results that may look feasible, but which are in fact not what the user wanted.

Selecting the Wrong Set of Solids

For each component involved in an Assembly Weight Management calculation, the calculation will involve the solids which are included in the weight reference set for that component. There is a tendency to think that it will use the solids which are visible, but visibility actually has no bearing on the calculation. It is therefore critical that the weight reference set is correctly set for every component in the calculation, otherwise incorrect results might be generated.

For example, the wrong results would be generated in the following two situations :

- Some components contain reference geometry but their weight reference set is still set to Entire Part. This causes the reference geometry to be included in the mass properties of the product. The implementation of appropriate standards can prevent this problem (see Standard Weight Reference Set - earlier in this chapter).

- The user creates reference sets of a standard name (eg. WEIGHT), but only sets the weight reference set to WEIGHT in the top level assembly. The WEIGHT reference set must be set in every component which contains solids, unless the default (Entire Part) is appropriate.

Selecting the Wrong Set of Components

The set of components used in an Assembly Weight Management calculation is determined either by the user selecting components, or by the current setting of the Assembly Weight Management Component Set in the work part (all components, by default).

It is not determined by the set of visible components, and it can include components that have been excluded by reference set.

The wrong results would be generated in the following two situations, for example:

- The user has loaded the top level assembly with no components and then has loaded all of the components in one of the subassemblies. They want to find the mass properties of the subassembly, and assume that by running an Assembly Weight Management calculation on the work

part, it will only use the loaded components. In fact it would use all of the components in the top level assembly, unless the Assembly Weight Management component set has been changed.

- The user has used subassembly reference sets to deliberately exclude some components (to see only those components which are used in a particular variant of a product, for example). To ensure that only the right components are included in the Assembly Weight Management calculation, they use the active subassembly reference sets as the weight reference set for those subassemblies. This would again generate the wrong results. The Assembly Weight Management reference set only determines which solids from that component are included in the calculation, it does not determine which lower level components are included.

Using an Out of Date Assembly

When using Assembly Weight Management for an assembly which does not have all of its components loaded, it is possible that out of date caches will be used. If you want to get the right result without loading all of the components, you should use the recommended procedure.

Promotions

If you include a promoted body in an Assembly Weight Management calculation, you must make sure that the base component for the promotion is also included. The technique that Assembly Weight Management uses to deal with promotions relies on this being the case.

Therefore when you use the Selected Components or Set Component Set buttons to specify a set of components which includes a promoted body, be sure to include the base component for the promotion. Otherwise incorrect results might be generated.

Access to Cached Results After Changing Work Part

Component caches are only available to the assembly in which they were created. Therefore, if you the change work part or displayed part during a session, different sets of cached results might be available at different times. This can sometimes cause confusion.

If you have generated the mass properties for an assembly and you want to inquire the mass properties of a subassembly:

- Instead of changing work part, use the Selected Components or Set Component Set buttons to specify the set of components you want to include in the calculation (i.e. all of those in the subassembly)

The first of these approaches is generally preferable as up to date component caches become available. Then you will not have to load any extra parts.

Low Accuracy Results

Occasionally a solid can have an unusually high error estimate. This means that NX is not confident that the results of its Assembly Weight Management calculation for that solid are close to the real values.
It is therefore a good idea to periodically check the error estimates. As mentioned in Appendix C, these estimates are sometimes over cautious and can be unduly alarming. However if the error estimate is particularly high, it is worth tracking down the components containing the solids which are causing the problem and increasing the accuracy setting for those components. The problem is most likely to occur with solids that have a very long, thin shape and which contain one or more faces with complex geometry. If you have problems with unexpectedly inaccurate results, look for this type of solid first.

Summary

This lesson concentrated on how to use Assembly Weight Management to optimize the process of monitoring the mass properties of large assemblies.

In this lesson you:

- Generated caches of analysis information on piece parts.
- Asserted the mass of a component with a known weight.
- Set a "Not to Exceed" weight limit on an assembly.
- Defined a reference set for calculations.

Lesson

7 Applications for Advanced Assemblies

Purpose

This lesson introduces potential applications for Advanced Assemblies and offers ideas for possible approaches to these applications.

There are essentially three different types of users that will interact with a large assembly. Each user has different requirements.

Objectives

Upon completion of this lesson, you will be able to:

- Apply Advanced Assemblies
- Identify the three types of users that will interact with large assemblies
- Identify the requirements of the three types of large assembly users

7

Three Types of Large Assembly Users

1. **Design-In-Context User**

   This user is interested in designing or editing a single part or a small group of parts that interface with, or physically fit into a larger assembly. The user likely will not have ownership or write access to the large assembly. The interfacing parts probably do not come from the same "group" or lie along the same assembly tree lines. The typical task required of the system will be to load all of the parts that are within 6 inches of my work part or a zone I have defined.

   The user will need the system to stay active or flexible enough so he can get the updates to the other parts as well as respond to any new requirement he might have.

2. **Assembly Design Reviewer**

   This person might be a lead designer or a supervisor in charge of reviewing others designs or an integration engineer in charge of verifying fit for the large assembly. Typically they are not responsible for designing geometry personally, but will validate or check fit and functions etc. The user will likely want to see specific "zones" and/or systems such as all electrical components below the floor. The user might prefer to evaluate the assembly via a sectional analysis in which they retrieve the assembly only loading the components that cross a given plane or belong to a certain zone.

3. **Assembly Documentation User**

   This user is responsible for producing documentation (pictures or drawings) of the assembly.

   This user is usually required to create several types of drawings. One type might be the entire assembly as a whole. Another type might show the installation of a subsystem in the large assembly showing portions of the large assembly for reference in phantom. Since the results of a drawing or picture are supposed to be aesthetically pleasing, it is likely the system will not provide the user with exactly the correct components that are desired. The user will probably need to do some manual opening and closing of some components to get the exact configuration desired.

Appendix

A Appendix A: Assembly Navigator

Purpose

The Assembly Navigator provides a graphical display of the assembly structure and a quick and easy method to manipulate components in an assembly.

Objective

Upon completion of this lesson, you will be able to:

- Activate the Assembly Navigator window.
- Use the pop-up and pull-down menu options in the Assembly Navigator window to manipulate the assembly

A

General Information

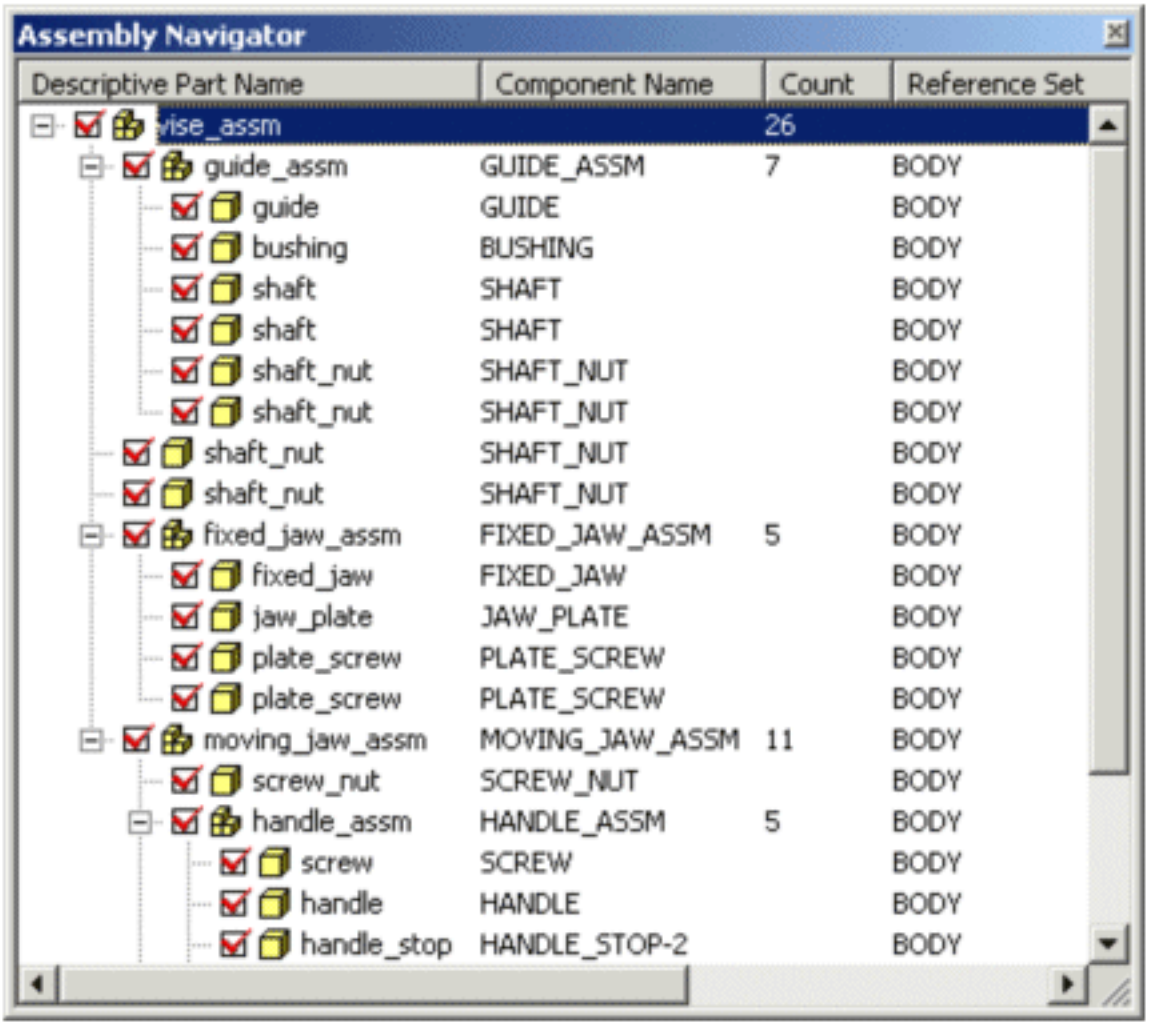
The Assembly Navigator, also know as the Assembly Navigation Tool (Assembly Navigator), gives you a graphical display of the assembly structure of the displayed part in a separate window, and provides a quick and easy method of manipulating components in an assembly. For example, you can use the Assembly Navigator to select components for various operations, as well as to perform assembly management functions such as changing the work part, changing the displayed part, blanking and unblanking components, and more. The Assembly Navigator represents the assembly structure as a graph of objects displayed as a tree. Each component is displayed as a node in the assembly tree structure.

Opening the Assembly Navigator

To open the Assembly Navigator, press the Assembly Navigator tab. (Unix users, choose the Assembly Navigator icon on the Navigators toolbar.).

The Assembly Navigator Window

When you start the Assembly Navigator, the Main Window is displayed, as shown below. You can either resize the display or use the scroll bars to see the entire tree structure.



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Node Display

Each component of an assembly is displayed as a node in the assembly tree structure. If you select on a node with MB1, the system will highlight the component geometry in the graphics window. Each node consists of a checkbox, an icon, and the part name, plus the other columns. If the part is an assembly or subassembly, an expand/collapse box will also be present. Components may be selected for various operations, (i.e. Assemblies→Change Work Part, Edit→Object Display, etc.), by choosing the appropriate node in the Assembly Navigator with Mouse Button 1.

Assembly Navigator Icons

You can determine the loading status of the assembly by the state of the assembly two structure icons. Multiple "block" icons denote assemblies or subassemblies


Fully loaded assemblies / subassemblies.


Partially loaded assemblies / subassemblies.


Unloaded assemblies / subassemblies.

Single "block" icons denote components of assemblies:


Fully loaded part.


Partially loaded part.


Unloaded part

A

Expand/Collapse Box

Children of a node are only displayed when it is expanded. To expand or collapse the node, place the cursor over the box and click MB1. When a node is collapsed, the expand/collapse box is marked with a +. An expanded node is marked with a -.

Checkbox

Provides a quick means of determining a part's status. A checkbox also lets you load and show a part with a single action.

You cannot close a part by clicking on its checkbox. To close a part, use the File→Close options or the Close options on the Assembly Navigator popup menu.

The part's status is shown by the checkbox indicator.

No Check

The part is closed. Clicking on this type of checkbox:

- Loads the component and its children partially or fully, depending on the load options. Unloaded parents may also be loaded at this time.
- Any components that were blanked are now unblanked.
- Afterwards, the checkboxes of the part and its children have large red checks, except for those whose loading failed, who have an excluded reference set, or are on invisible layers.

A Thin Gray Check

The part is blanked, and at least partially open. It also appears for unblanked parts which either have an excluded reference set or are on invisible layers. Clicking on this type of checkbox:

- Unblanks the component, along with any of its children that were blanked.
- If any of its children were unloaded, they are now loaded.
- Afterwards, the checkboxes of the part and its children have large red checks, except for those whose loading failed, who have an excluded reference set, or are on invisible layers.

A Large Red Check

The part is unblanked, at least partially open, in a visible reference set, and on a visible layer. Clicking on this type of checkbox:

- Blanks the component and its unblanked children.

- Afterwards, the component's checkbox has a thin gray check, and its children have gray checks (if blanked) or no checks (if unloaded).

Selecting Components

When you can select components using a dialog, you can also select them using the Assembly Navigator by choosing the appropriate node with MB1. You can select single or multiple components. When selection is complete, you can process the components in the list by choosing OK from the dialog. To select multiple components in the Assembly Navigator, select the first component and then either:

- Use <shift> MB1 (together) on another component to select all the components between those components

- Or use <ctrl> MB1 on another component if you want only it and the first component

  You can also use <shift> MB1 or <ctrl> MB1 on selected components to deselect them.

Identifying Components

If you click MB1 while the cursor is over a non-work part whose checkbox has a red check, that part is highlighted. The highlighting stays until you select another part. (Hovering the cursor without clicking MB1 has no effect.) If you hover the cursor over a part that is not visible (e.g., blanked, on another layer, out of the reference set, unloaded), a box defining the boundaries of the component appears in the graphics window. The box disappears when you move the cursor to another part. This only happens when the Preselect Invisible Nodes preference is toggled ON. Checkboxes for parts that are not visible will have a thin gray check or no check.

Because of configuration differences, you may have to hold MB1 down for a couple of seconds before the box is drawn. In some cases, the box may not be drawn until you release MB1. Also, the box will not be drawn if you double-click MB1.

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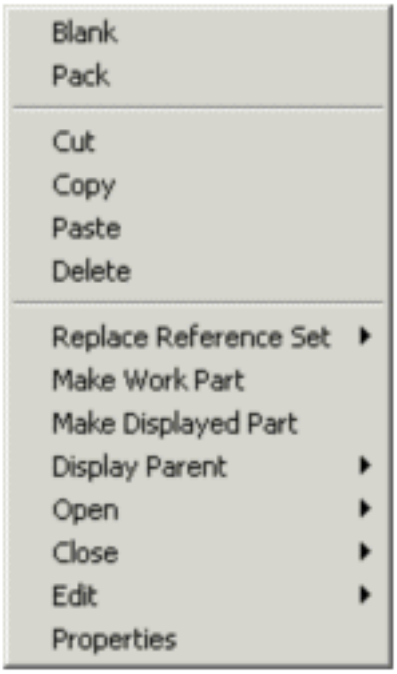
Pop-Up Menu Options

If you position the cursor over a node in the tree that represents a component and click MB3, a popup menu with the following options appears. Depending on the type of component; all options may not be available.



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Blank / Unblank

Blank or unblank components and subassemblies. (Blanking a subassembly causes it and all its components to be removed from the graphics display. Unblanking a component restores it to the graphics display.)

Pack or Unpack

Pack removes multiple occurrences from the Assembly Navigator display and replaces each multiple occurrence with a single icon. (Multiple occurrences are components with the same parent, and whose prototype is the same part.) Unpack reverses this process; it lets you show all occurrences.

Make Work Part

Selects the part in which to create geometry, giving you the ability to design in context.

Make Displayed Part

Switches the display between currently loaded parts. The displayed part becomes the top node of the Assembly Navigator display.

Display Parent

Switches the display from a component to one of its parent assemblies, including the Start Part if the current node represents a linked part.

Replace Reference Set

Replaces the display of a component in the graphics screen with one of its reference sets.

Open

Component
Component As Child
Components Assembly
Assembly As Component
Fully
Opens various components within the assembly structure.

Close

Part
Part (Modified)
Assembly

Closes the selected component (the Part option) or the entire assembly (the Assembly option). If the part has been modified, the Part (Modified) option will be active to alert you that you might wish to first save the file. Once you have closed the component part, you can later reopen it using the Open option. If you selected multiple components, Part closes only the components that are not modified, while Part (Modified) closes all of them.

Select Assembly

Lets you select the component and all its children.

Properties

Brings up the Component Properties dialog which has 5 tab screens:

- Assembly
- General
- Attributes
- Weight
- Part File

Activity: Activating/Using the Assembly Navigator

In this activity, you will explore the functionality available when using the Assembly Navigator.

Step 1: Open dau_hole_punch_assm from the appndx_a directory and start the Modeling application.

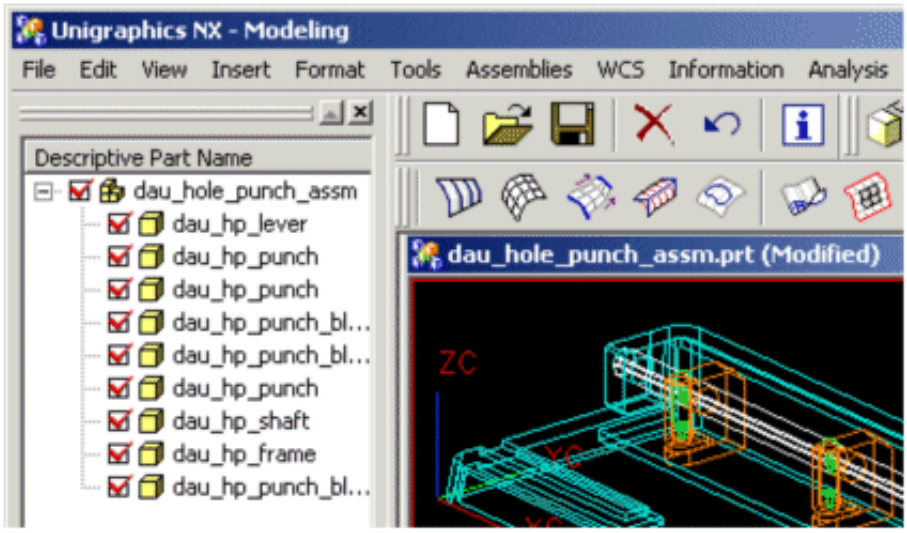
Step 2: Activate the Assembly Navigator

☐ Choose the Assembly Navigator icon on the Resource bar (Unix users, choose the Assembly Navigator icon on the Navigators

Step 3: Dock the Assembly Navigation Window

Docking the Assembly Navigator allows the system to locate the Assembly Navigator window and resize the graphics window automatically.

Step 4: Drag the Assembly Navigator window to the left side of the NX Modeling window. The Assembly Navigator docks itself to the left side of the NX Modeling window.



Step 5: Click on the **X** in the top banner of the Assembly Navigator. The Assembly Navigator window disappears.

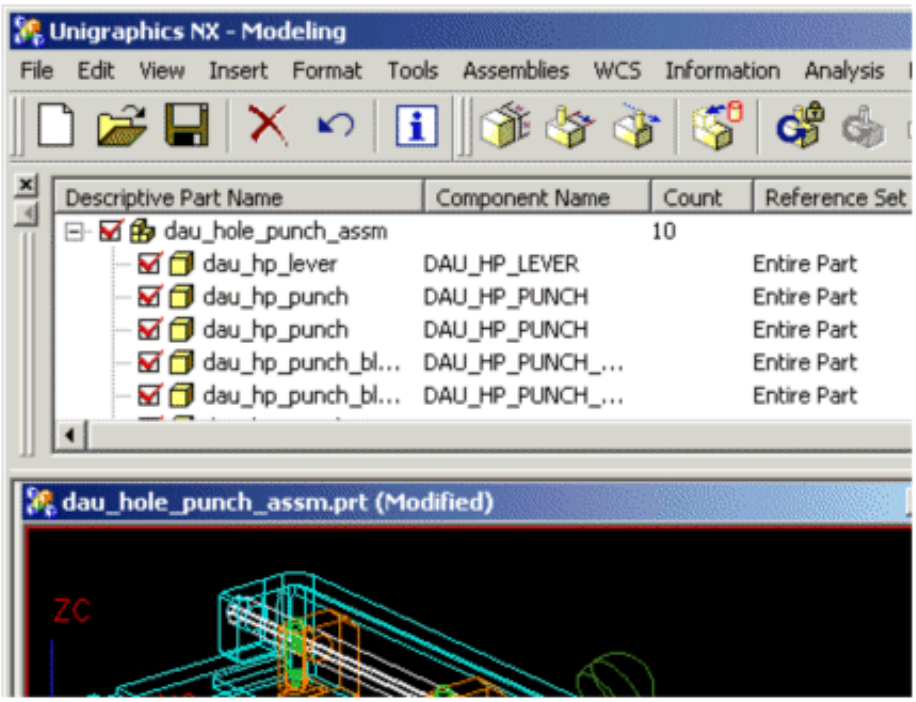
Step 6: Double-click on the Assembly Navigator tab on the Resource bar (Unix users, choose the Assembly Navigator icon on the Navigators toolbar). The Assembly Navigator window reappears in its docked location

A

Step 7: Select the double bars on the top of the Assembly Navigator and drag it into the center of the display to undock it.

☐ Again, select the double bars and drag the Assembly Navigator window (watching the "ghost" outline) to a horizontal position below the icon bars at the top of the display.

The Assembly Navigator docks below the bottom row of icons in the NX Modeling window.



Step 8: Verify the structure of the assembly

☐ Select various nodes in the Assembly Navigator, notice the associated component geometry highlights in the graphics screen.

Step 9: Use the Assembly Navigator to select components when the Change Work Part dialog is active.

☐ Choose Assemblies→Context Control Set Work Part.

☐ From the Assembly Navigator window, select one of the dau_hp_punch_block nodes.

☐ Choose OK in the Change Work Part dialog.

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When a component selection dialog is active, the component may be selected from the Assembly Navigator as well as the list and graphics screen.

Step 10: In the Assembly Navigator, double click on the dau_hole_punch_assm node to make it the Work Part

Step 11: Pack a component that has multiple occurrences.

☐ Place the cursor over one of the dau_hp_punch nodes.

☐ Press Mouse Button 3 and choose Pack from the pop-up menu.

Step 12: Blank and Unblank components.

☐ Using the mouse button 3 pop-up menu, Blank the *dau_hp_lever*

Note that the dau_hp_lever component geometry will disappear from the graphics display.

Step 13: Using the Mouse Button 3 pop-up menu, Unblank the dau_hp_lever

Blanking and unblanking can also be accomplished by simply clicking on the checkmark. 

Step 14: Change the Displayed Part and the Work Part using the Assembly Navigator pop-up menu.

☐ Using the Mouse Button 3 pop-up menu, make the dau_hp_frame the Displayed Part.

Step 15: Change the Displayed Part back to the hole_punch_assm.

☐ Place the cursor over the node in the Assembly Navigator.

☐ Press MB3 and choose Display Parent→dau_hole_punch_assm

Step 16: Close all parts

Specifying Columns

The Assembly Navigator uses columns to organize information. The number of columns depends on the assembly model opened. You can dynamically control this.

If you use assemblies with UG/Manager, these additional columns display:

- Part Number
- Part Revision
- Non-master types
- Non-master dataset name
- Item type
- Sequence number

You can modify these columns through the Assembly Navigator window by placing your cursor in the gray bar below the Assembly Navigator banner.

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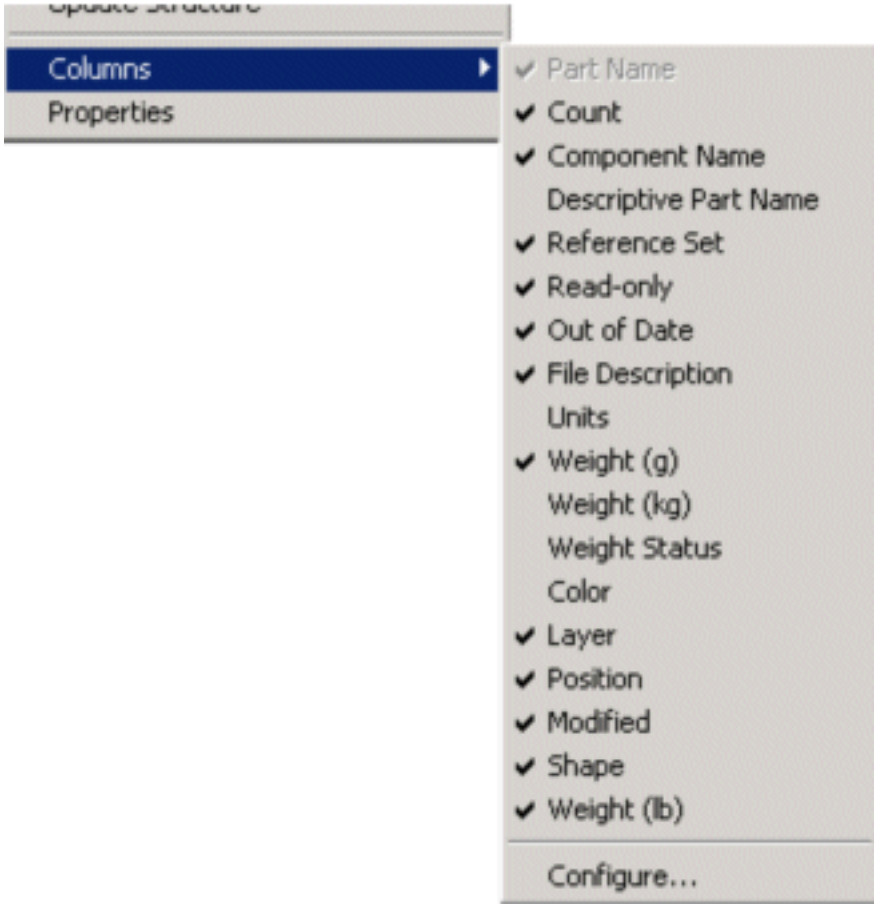
Once your cursor is positioned, you can use MB3→Columns to see the current configuration.



The pull-down shows the available columns and which ones are enabled (checked) to be displayed. There is also method of configuring the order in which you see the columns in the Assembly Navigator. You can use MB3→Columns→Configure to get the Assembly Navigator Properties dialog.

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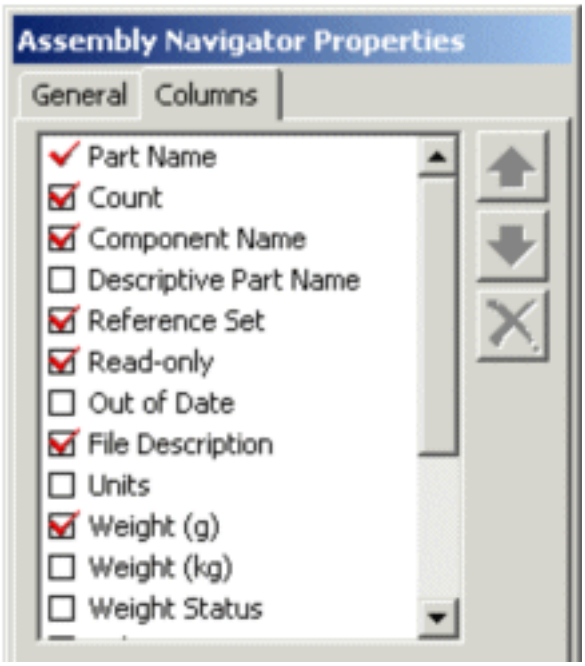
The Assembly Navigator Properties dialog appears.



In the Assembly Navigator Properties dialog you can turn columns on and off and also move the various column designations up or down to change the order in which they appear in the Assembly Navigator.

Certain columns have specific symbology to designate the respective status of components.

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Summary

The Assembly Navigation Tool provides a graphical display of an assembly which allowed you to quickly and easily select components and manipulate the assembly. In this lesson you:

- Activated the Assembly Navigator for an existing assembly.
- Interpreted the display of the nodes in the Assembly Navigator window.
- Used the pop-up and pull-down menu options to perform operations on components in the assembly.

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Appendix

B Appendix B: Command Line Batch Submission

Batch Command Line Options

You can run a batch clearance analysis program from the command line as an external User Function program. You must specify the executable to run, along with the following processing options:

ug_clearance:

$UGII_ROOT_DIR/ug_clearance <option> <partfile>

-c(olor)=<color> Name of valid color assigned to interference objects.

-h(elp) Request batch clearance analysis help message be output.

-l(ayer)=<layer>Layer on which interference objects are to be placed.

- m(ode)=<solid | facet_solid | full_facet | quick_facet>Specifies the clearance analysis mode.

-n(ame)=<dataset>Name of the Clearance Analysis Dataset to be analyzed.

- r(eport)=<log file>Name of the output report file.

-w >Request that mail be sent on completion of analysis.

-z(ones)=<x,y,z> Specify number of zones in the X, Y, and Z directions.

- pim=<yes|no> Run in IMAN or UG/Manager mode.

- user=<name>IMAN or UG/Manager user name.

-password=<name>IMAN or UG/Manager password.

-group=<name>IMAN or UG/Manager group name.

On VMS systems the above options are specified as /color or /c, etc.

part file =Specifies a part file on which to perform analysis. This file can be an assembly. The part is loaded using the load options stored in the load_options.def file from the current directory. When the analysis is complete, the modified part file is saved under the same name.

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Appendix

C Appendix C: Assembly Weight Management Accuracy Setting

For some solids containing complex geometry, mass property values cannot be calculated to absolute accuracy. In these cases an approximation technique is used for the calculation. The accuracy can be varied and is controlled by the Assembly Weight Management accuracy setting. By increasing the value of the accuracy setting, the calculation becomes more accurate but takes longer to complete.

The Assembly Weight Management menu allows you to assign an accuracy setting of 0.9, 0.99, 0.999, 0.9999, 0.99999, or 0.999999. This represents a practical spectrum of useful settings, ranging from low to very high accuracy. The values are equivalent to the same accuracy values in the Analysis→ Mass ... menus.

The approximation method involves NX making a first guess calculation. The calculation is then repeated with increasing accuracy, converging on the exact answer until the difference between two consecutive estimates, expressed as a fraction of the calculated answer, is less than (1.0 -accuracy).

For example if the accuracy setting is 0.99 and the weight is calculated at 5kg, the calculated value will be judged to be accurate enough when :

difference between last 2 calculations (in kg)/ 5 < 0.01

In other words the calculation would be accurate enough when the difference between consecutive calculations is less than 0.05 kg.

Normally the difference between the final calculation and the exact mass is less than the difference between the last two calculations. This would mean that the Assembly Weight Management calculation above is likely to give an answer within 0.05 kg of the genuine weight.

If you set the Give Error Information option, NX will list an estimated error range for the values it calculates. These error estimates are often very cautious. If an error estimate for a part appears to be high, it is best to repeat the calculation with a slightly higher accuracy setting. If the results of the weight calculation are very similar to the last one, the first error estimate was probably overcautious.

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