

Introduction to NX for Experienced Users

**Student Guide
August 2010
MT13155_S — NX 7.5**

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Course overview

Intended audience

This course is suited for designers, engineers, manufacturing engineers, application programmers, NC programmers, CAD/CAM managers, and system managers who need to manage and use NX.

Prerequisites

There are no prerequisites for this class.

Course objectives

After successfully completing this course, you should be able to:

- Open and examine NX models.
- Create and edit parametric solid models.
- Create and modify basic assembly structures.
- Create and modify simple drawings.

Lesson format

The general format for lesson content is:

- Instructor presentation
- One or more activities
- Summary

Learning tips

- Ask questions.
- Confirm important facts by restating them in your own words.



It is important to use your *Student Guide* in the sequence it is written.

Common symbols

The student manual uses special symbols as shown below.



Design Intent — Information about the task and what must be accomplished.



Tip — Useful information or advice.



Note — Contains useful information that supplements or emphasizes the main points.



Example — Shows a possible way that the current topic of discussion could be used.



Caution — Contains important reminders or information about a task.



Warning — Contains information essential to your success.

NX 7.5 Help Library

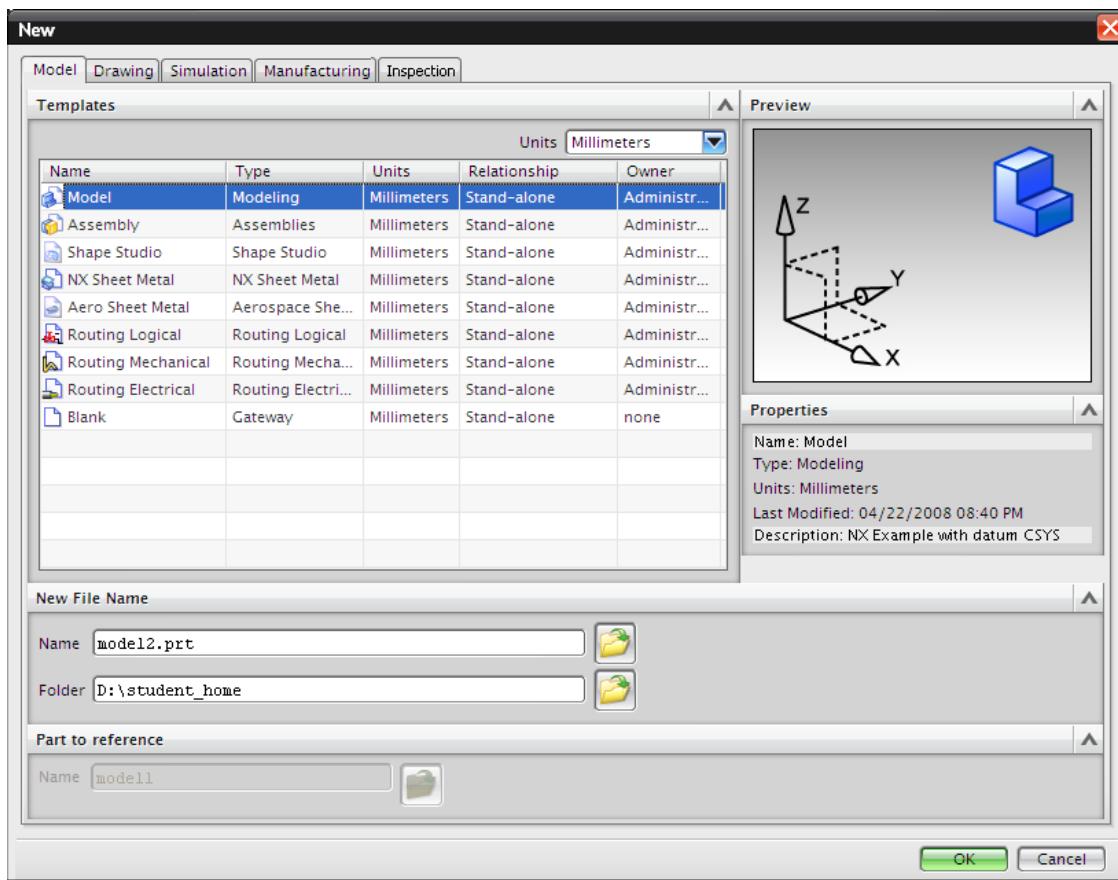
The NX 7.5 Help Library is available online any time you need more information about a function. To access the NX 7.5 Help Library; from the NX menu bar choose **Help→Documentation**.

Template parts

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

- A frame of reference, such as a datum coordinate system
- Commonly used expressions
- An initial application such as Modeling, Drafting, or Sheet Metal
- Part attributes, for example, attributes for a parts list
- Drawing formats
- User-defined views
- Layer categories

Choose a template from the New dialog box.



Teamcenter Integration for NX vs. native NX terminology

Teamcenter Integration for NX Term	Native NX Term
Item	Part
Item revision	Part revision
Dataset	Part file
Item ID	Part number
UGMASTER dataset	Master part file
UGPART dataset (specification or manifestation)	Non-master part file (for example, a drawing or manufacturing file)

When you work in NX, you manipulate parts, part revisions and part files. These correspond to items, item revisions, and datasets in Teamcenter Integration for NX and Teamcenter.

Layer standards

Parts used in this course were created using layer categories the same as or very similar to those found in the **Model** template parts.

Layers provide an advanced alternative to display management (Show and Hide) to organize data.

Layer categories in the Model template parts

Layers	Category	Description
1–10	Solids	Solid bodies
11–20	Sheets	Sheet bodies
21–40	Sketches	All external sketches
41–60	Curves	Non-sketch curves
61–80	Datums	Planes, axes, coordinate systems
81–255	No category assigned	

Implementing a layer standard

You may implement or enforce layer standards using some of the methods below:

- Create NX Open programs to create a standard part organization and verify it upon release.
- Use a macro to create layer categories: **Tools**→**Macro**→**Playback**.
- Your administrator can enforce company standards by providing suitable templates.



In this course you *may* use a layer organization method you anticipate using in your work.

Student responsibilities

- Be on time.
- Be considerate of the needs of other students.
- Listen attentively and take notes.
- Ask questions.
- Practice what you learn.
- Have fun!

Lesson

1 NX part files

Purpose

This lesson is a fundamental introduction to working with NX part files.

Objectives

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

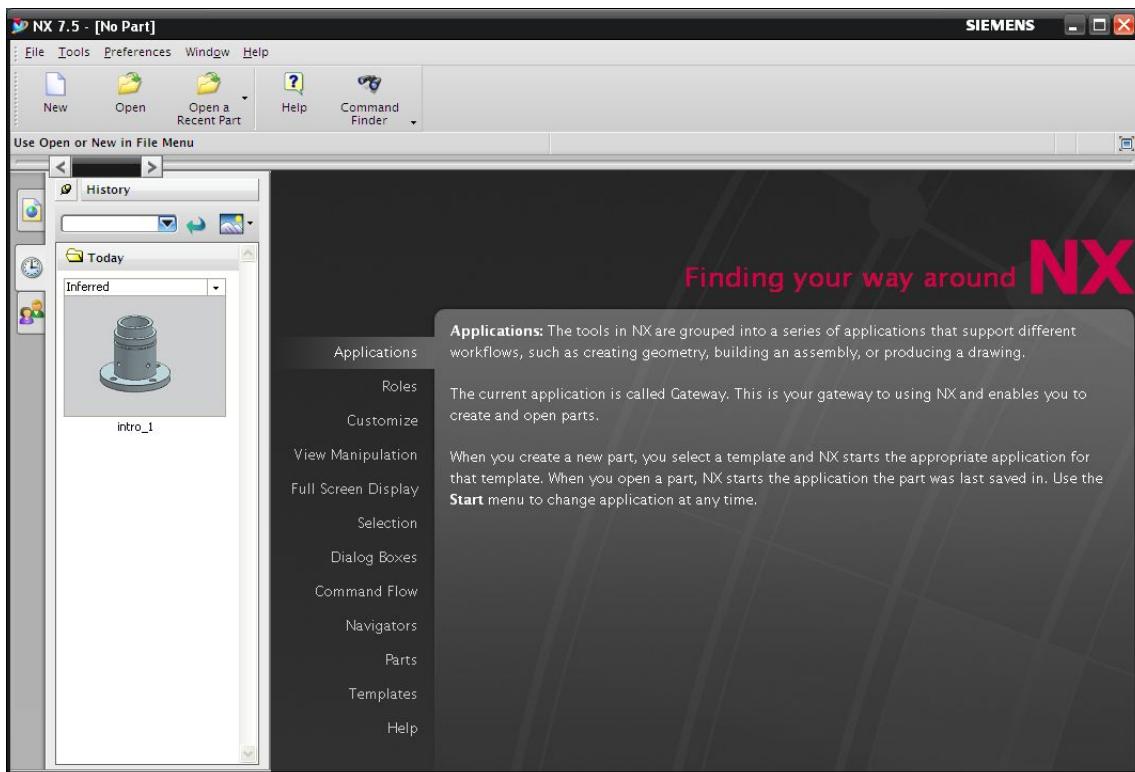
- Start an NX session.
- Create a new part file.
- Open a part file.
- Copy a part file.
- Close a part file and exit NX.

Introduction to NX

The first step in working in NX is to log on to a workstation and start an NX session.

- Your instructor will provide the steps needed to log in and start NX in the classroom.

After you start NX, you see the **No Part** interface. You can change defaults and preferences, open an existing part file, or create a new part file.



Gateway application

The tools in NX are grouped into a series of applications that support different major workflows, including creating geometry, building an assembly, or producing a drawing.

Gateway is the first application you access when you:

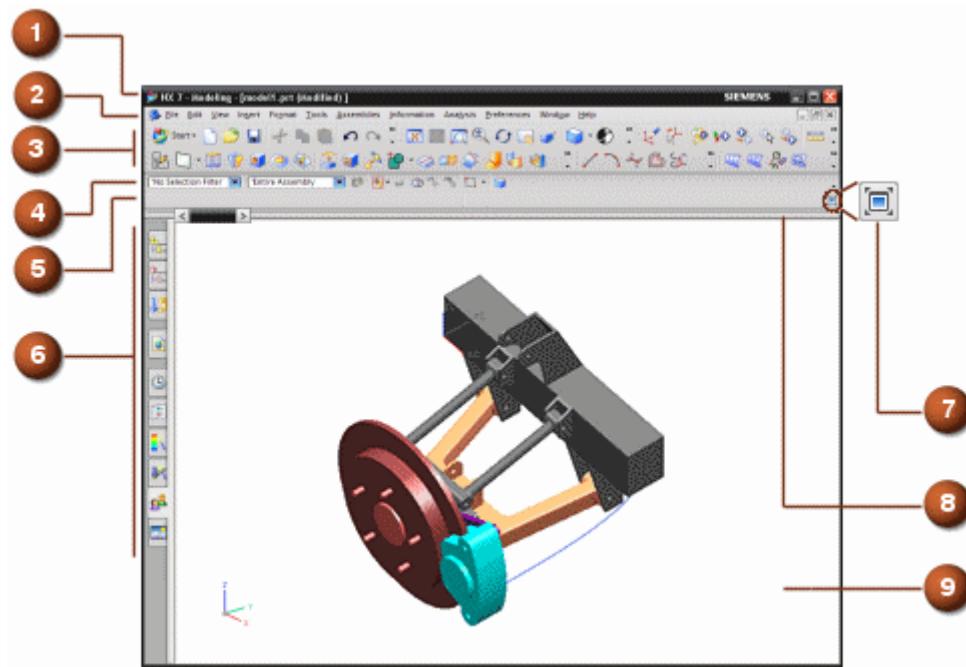
- Create a new blank part file.
- Open a part file that was saved in Gateway after NX 4.
- Open a part file that was last saved in NX 3 or earlier.

Gateway allows you to review existing parts.

To create or edit objects within a part, you must start another application, such as Modeling.

The NX Window (standard)

In *standard display*, the menus and toolbars are positioned at the top of the window and the Resource bar is to the left of the graphics window. This is the default display.



#	Component	Description
1	Title bar	Displays information for the current part file.
2	Menu bar	Displays menus with lists of commands.
3	Toolbar area	Displays active toolbars.
4	Selection bar	Sets selection options.
5	Cue and Status line	Prompts you for the next action that you need to take, and displays messages on functions and actions.
6	Resource bar	Contains tabs for navigators, browsers, and palettes.
7	Full screen button	Lets you switch between standard and full screen displays.
8	Dialog Rail	Positions dialog boxes.
9	Graphics window	Lets you create, display, and modify parts.



New file overview

Use the New command to select a template and create a new product file.

- Standard templates are available and grouped by types, such as modeling, drawing, simulation, and manufacturing.
- Use blank templates to create files with no custom content.
- When you create a new file from a template, it has a copy of all the objects in the template and inherits all its settings.
- Your system administrator can create customized templates based on the requirements at your site.

After you create the file, NX starts the appropriate application based on the template. For example, if you select a modeling template, NX will start Modeling.

A default name and location for the new file is assigned based on customer default settings for each template type.

You can change the name and location:

- Before you begin work on the file.
- In native mode only, when you save the file for the first time.

You can specify a master part to reference when you create a new non-master file.

Where do I find it?

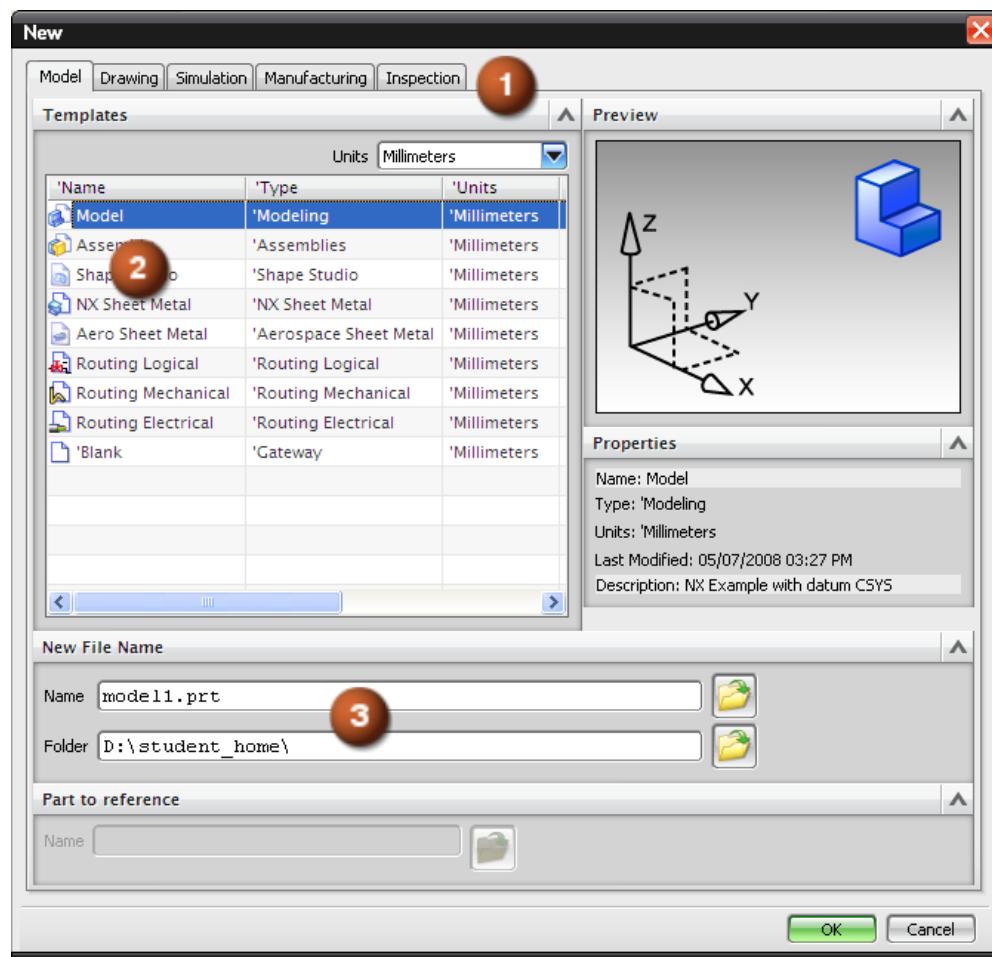
Application	Gateway
Toolbar	Standard→New 
Menu	File→New

Use a template to create a new file

1. On the **Standard** toolbar, click **New** .
 2. Click the tab for the file type you want (1).
 3. Select the template you want (2).
 4. Type or assign the number and revision information (3).
- (Optional) Type the name and path information (3).



You can also type this information when you save the file.



Layers

Use layers to organize geometry.

Use layer categories to organize and name layers.

Choose **Format**→**Layer Settings** to access the **Layer Settings** dialog box.

There are 256 layers available in an NX part file, one of which is always the work layer.

The display characteristics of layers are controlled by the **Name** and **Visible Only** columns of the **Layer Settings** dialog box.

Name	Visible Only	Object Count	Categories
1 (Work)		3	Solids
<input checked="" type="checkbox"/> 2	<input checked="" type="checkbox"/>	1	Solids
<input type="checkbox"/> 21	<input type="checkbox"/>	2	Sketches
<input checked="" type="checkbox"/> 62	<input type="checkbox"/>	5	Datums
<input checked="" type="checkbox"/> 61	<input type="checkbox"/>	2	Datums

- **Name** – This column shows the active **Work** layer (the layer on which objects are created). Layers other than the work layer each have a corresponding check box which controls layer visibility and the ability to select objects on that layer (a layer is visible and selectable if its check box is checked).
- **Visible Only** – This column has a check box for each layer other than the Work layer. If a check box in this column is selected, the objects on that layer can be seen but not selected.

The work layer is the layer that objects are created on and is always visible and selectable.

When you create a new part file, layer 1 is the default work layer.

When you change the work layer, the previous work layer automatically becomes selectable. You can then assign it a different status.

The number of objects on one layer is not limited. You may choose which layers to create objects on and what the status will be.

Layer categories in the Model template parts

Layers	Categories	Description
1–10	Solids	Solid bodies
11–20	Sheets	Sheet bodies
21–40	Sketches	All external sketches
41–60	Curves	Non-sketch curves
61–80	Datums	Planes, axes, coordinate systems
91–255	No category assigned	

Activities: NX part files — Create new

In the *NX part files* section, do the activity:

- *Create new part files*



Open file overview

Use the Open command as an alternative to the Teamcenter Navigator to open an existing product file.



NX part files have a *.prt* extension.

When a part file is open:

- The graphics window shows the model in the condition in which it was last saved.
- The title bar of the graphics window displays the name of the current work part.
- If the part is read only, the words **Read Only** appear beside the part name. This means that changes may not be saved in this file.
- If an Application was not *already active*, NX starts the application in which the part was saved.

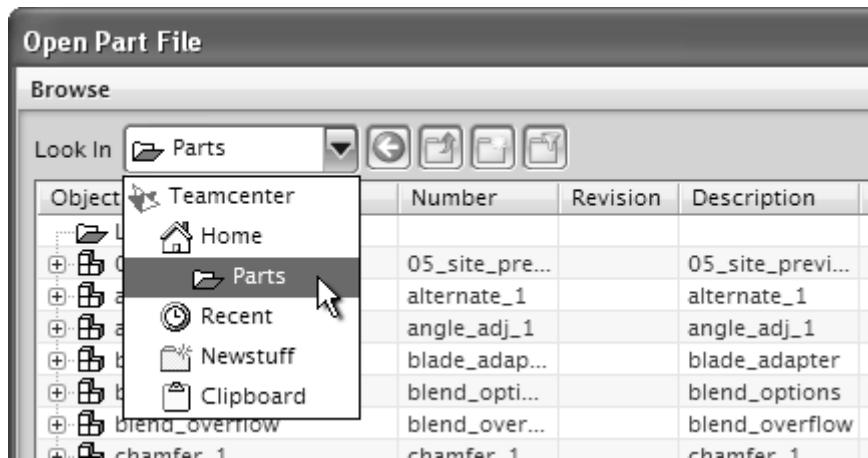
A loaded part is only a copy of what is stored on disk. Any new work that you do is not permanent until the part is saved.

Where do I find it?

Application	Gateway
Toolbar	Standard→Open 
Menu	File→Open

Useful features on Windows file dialog boxes

The **Look in:** list shows the name of the current selected drive or folder.



Up One Level works with the **Look in:** option menu to traverse back up through the folder hierarchy.

Create New Folder option allows new sub-folders to be created in the current folder.

View Menu allows the appearance of the listing in the window to be modified.

Open multiple parts

You can open or load more than one part at any time and work on several parts concurrently.

There are two identifiers for loaded parts:

Displayed The part is displayed in the graphics window.

Work The part is accessible for creation and editing operations.

Change the displayed part

You can have multiple parts open, or *loaded*, at the same time.

Control which part is displayed in the graphics window by using **Window** on the menu bar.

The **Window** option works in two ways:

- Select a part from the list to display. The list contains up to ten recently displayed parts.
- Select **More** to display the **Change Window** dialog box.

The **Change Window** dialog box contains a list of all components in an assembly structure as well as any loaded parts not contained in a loaded assembly.

Save As

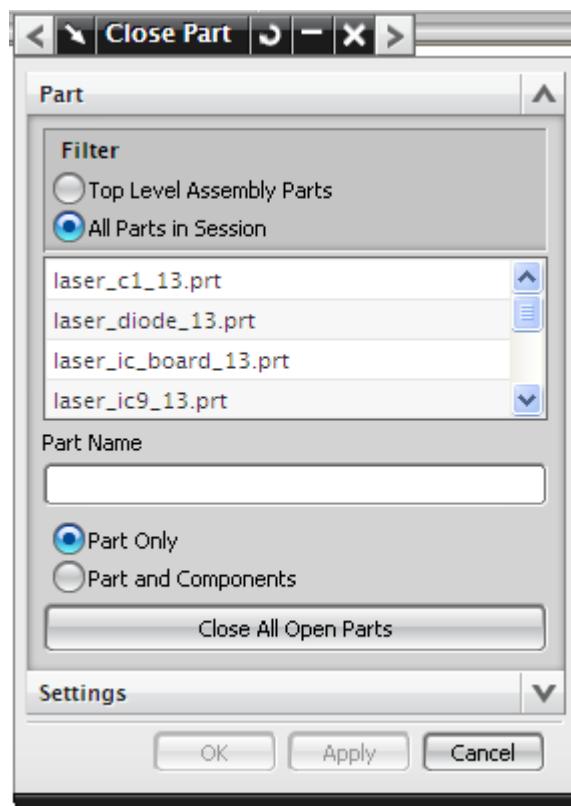
File→Save As allows you to save the current part under a different name and/or in a different directory.

When you select **Save As**, a file selection dialog box displays asking for the new name and location.

The name/location must be unique within the current directory. If you specify a name that already exists, an error message displays. The current part is filed under the new name, and the new part file name displays on the graphics window.

Close selected parts

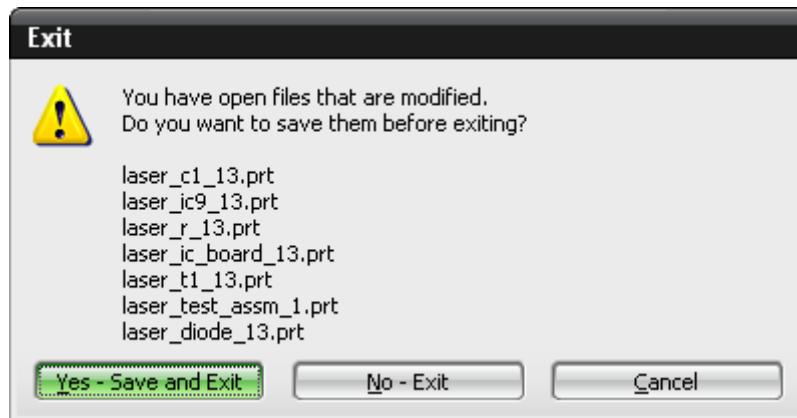
1. Choose **File**→**Close**→**Selected Parts**.
2. In the **Close Part** dialog box, select parts to close from a list.
3. Click **OK**.



Exit NX

End an NX session by choosing **File→Exit**.

If you modified any parts and did not save them, you get a warning message.



Activities: NX part files — Open, save, and close

In the *NX part files* section, do the activity:

- *Open, save, and close existing part files*

Summary: NX part files

In this lesson you:

- Started an NX session.
- Created, opened, and saved part files.
- Copied a part file.
- Closed a part file.

2 The NX user interface

Purpose

This lesson introduces the NX user interface.

Objectives

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

- Customize toolbars.
- Save and restore toolbars by applying a role.
- Select objects in the graphics window.
- Manipulate the orientation of the work view.

Customize and display toolbars

- Each application has its own set of toolbars. You can *hide* or *display* available toolbars for each application.
- You can either *display* or *hide* available buttons for each toolbar.
- For each toolbar you can *add* buttons from other toolbars, or *remove* them.
- You can save and share toolbar arrangements for all or selected applications, using **Roles**.

Docking toolbars

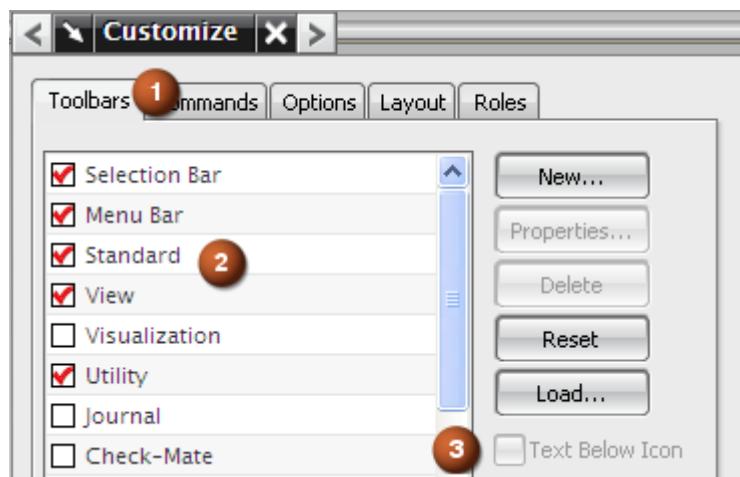
- You can dock toolbars horizontally or vertically in the NX window.
- You can move undocked toolbars on your screen.

Display toolbars

1. Choose **Tools**→**Customize** from the main menu bar.
2. On the Toolbars (1) page, select check boxes (2) to display toolbars and clear to hide them.



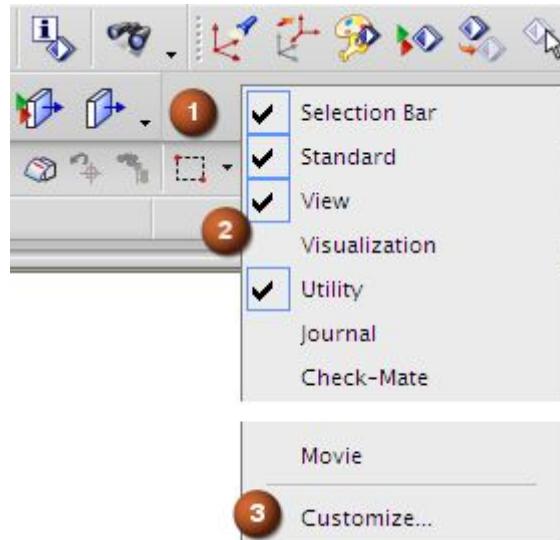
Select **Text Below Icon** (3) to display names on the buttons.



Display toolbars using the shortcut menu

1. Right-click anywhere in the toolbar area (1) to display a shortcut menu of all toolbars.
 2. Select the listed toolbar names to display toolbars or clear the check boxes to hide them (2).
-  Empty check boxes are *not* displayed beside menu items that are not selected.

 You can also select **Customize** (3) to open the **Customize** dialog box.



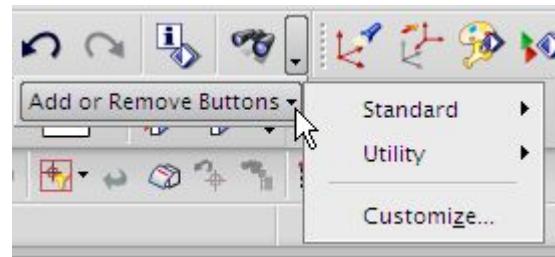
Add or remove toolbar buttons

Toolbar options are an efficient way to turn on and off the display of buttons within a toolbar.

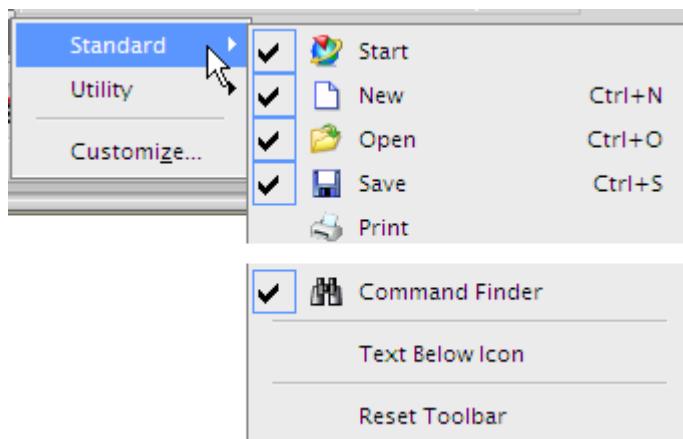
1. Click **Toolbar Options** on a toolbar and select **Add or Remove Buttons**.



2. Select a toolbar to modify, or select **Customize** to open the **Customize** dialog box.

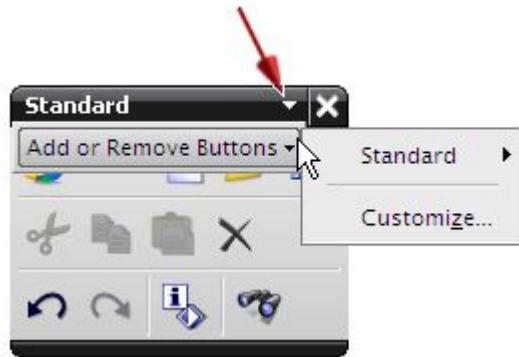


3. Click an item with no check box to display it. Clear the check box to hide an item.



Access options on undocked toolbars

Access toolbar options on undocked toolbars as shown below.





Command Finder overview

Use the **Command Finder** command to find and activate a specific NX command that is associated with one or more words or phrases that you enter. This includes commands that may not be active in the current application or task environment.

From the list of commands you can:

- Display the command location, when it is available in the current environment.
- Launch the command, if it is available.
- Turn on a toggle command, when it is available in the current environment.
- Access the Help information for the command.



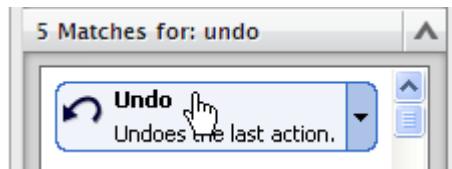
Where do I find it?

Application	Gateway
Toolbar	Standard→Command Finder
Menu	Help→Command Finder

Search for a command

1. On the **Standard** toolbar, click **Command Finder** , or choose **Help→Command Finder**.
2. In the **Search** box, type one or more words or phrases.
3. Click **Find Command**  or press Enter.
4. Place the cursor over any command presented in the **Matches** for list.

2



If the command is available for immediate use, the correct menu path or toolbar button is highlighted.



5. (Optional) Click any available command in the list to immediately activate it.
6. (Optional) Right-click a command in the list and choose Help to display additional information about the command.

Dialog boxes and the Dialog Rail

Dialog boxes that open when you choose an NX command are positioned by default on a Rail Clip which slides on a Dialog Rail.

To position the Rail Clip along the Dialog Rail, either drag the center of the Rail Clip or click the arrows to move it to default positions.



Dialog boxes are organized into groups that can be collapsed or expanded as needed. These groups contain different types of information and options

The typical workflow is to interact with the dialog box from the top to the bottom.



The three buttons along the bottom of the dialog box are, from left to right, **OK**, **Apply** and **Cancel**.

OK – Accepts all dialog box parameters and settings and closes the dialog box. Some dialog boxes use <OK> which implies an **OK** if another command is selected.

Apply – Applies the all dialog box parameters and settings but leaves the dialog box open for further input.

Cancel – Closes the dialog box without processing the dialog box parameters and settings.

If you need to see behind the dialog box, either slide the Rail Clip to either side or click the center of the Rail Clip to temporarily hide the dialog box. Click again to show it.

Dialog Rail buttons

Consistent options appear on the Rail Clip or on the dialog box title bar when the dialog box is not clipped to the Dialog Rail.



Rail Clip buttons

	Move Left	Moves the dialog box along the Dialog Rail to predefined positions.
	Move Right	
	Clip	Clips or unclips the dialog box to the Rail Clip.
	Unclip	When a dialog box is clipped, you can position it by sliding the Rail Clip along the Dialog Rail or by clicking the arrows to move the Rail Clip to predefined locations.
		When it is unclipped, the dialog box floats, and you can position it anywhere on the screen by dragging its title bar.
	Reset	Resets dialog box input values to the default values. When editing a feature, the default values are the values used when the feature was created.
	Hide Collapsed Groups	Shows or hides all dialog box groups that are currently collapsed.
	Show Collapsed Groups	This simplifies the display of the dialog box.
	Close	Closes the dialog box.



Roles

As you define your own roles, you or your administrator can add them to a palette for others to share.

Roles let you control the appearance of the user interface in a number of ways. For example:

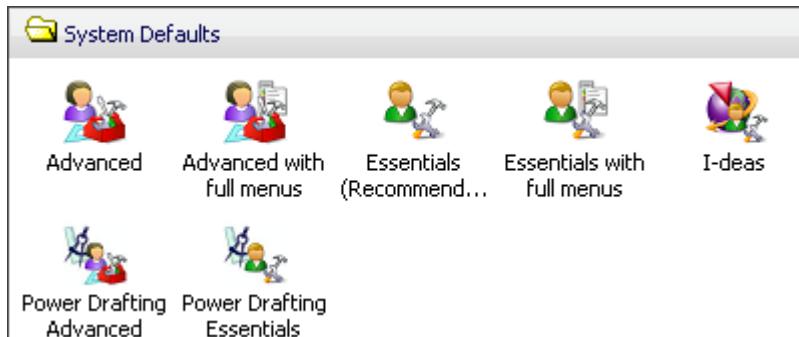
- The items displayed on the menu bar
- The buttons displayed on the toolbars
- Whether button names are displayed below the buttons

Example roles

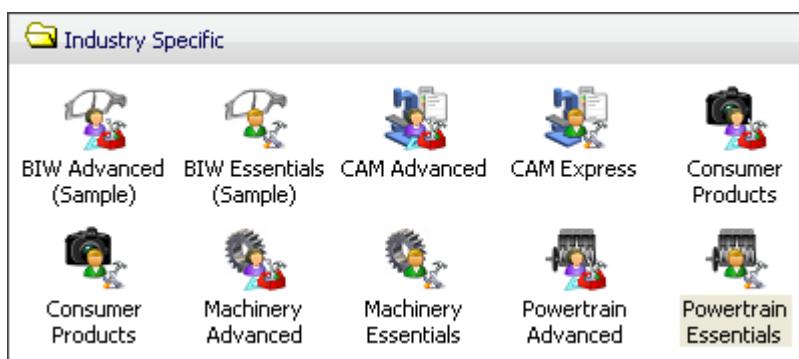
NX comes with a number of example roles. These give you a choice of starting points as you customize toolbars to meet your needs.

The roles palette includes these groups:

- **System Defaults** — generic roles for new and advanced users



- **Industry Specific** — examples of configurations for various industries



- **User** — exists after you save one or more personal configurations



For those starting to use NX or those who use NX infrequently, one of the **Essentials** roles in **System Defaults** is recommended.

For more information about any role, hold your cursor over its button.

Choose a role

1. In the Resource bar, click the **Roles** tab to display the palette.
2. Click the role you want or drag it into the graphics window.
3. Click **OK** to accept the new role.

Save toolbar configuration between sessions

When you exit an NX session, the current state of your toolbars is saved by default. They will be the same when you start a new session.

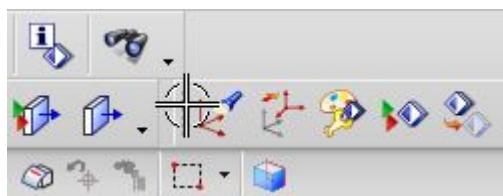
You can control how this is saved:

1. Choose **Preferences→User Interface**.
2. On the Layout page, select **Save layout at exit**.

Activities: User interface — Toolbars and Roles

In the *NX user interface* section, do the activities:

- *Customize toolbars*

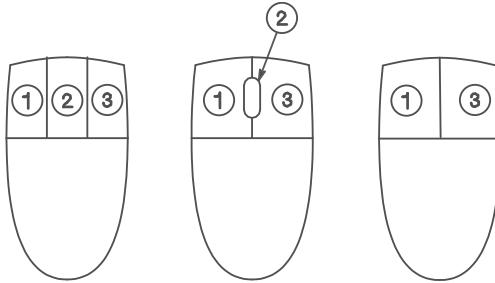


- *Create a new user role*



Using the mouse

There are three mouse configurations in common use.



On a two-button mouse, use the left (1) and right (3) buttons together when you need the middle button.

On a three-button mouse, you can use *combinations* of mouse buttons.

- Use middle (2) plus right (3) buttons to pan.
- Use middle (2) plus left (1) buttons to zoom.

Mouse Button	Action
	Select or drag objects.
	Click OK while in an operator. Press and hold down while in the graphics window to rotate the view. Hold down Shift and the middle mouse button to pan. Hold down Ctrl and the middle mouse button to zoom in or out.
	Display shortcut menu with various functions. Also display action information for currently selected objects.
Rotating mouse wheel	Zoom in and out in graphics window. Scroll in lists, menus, and the Information window.

Here is a summary of things you can do by moving the mouse cursor.

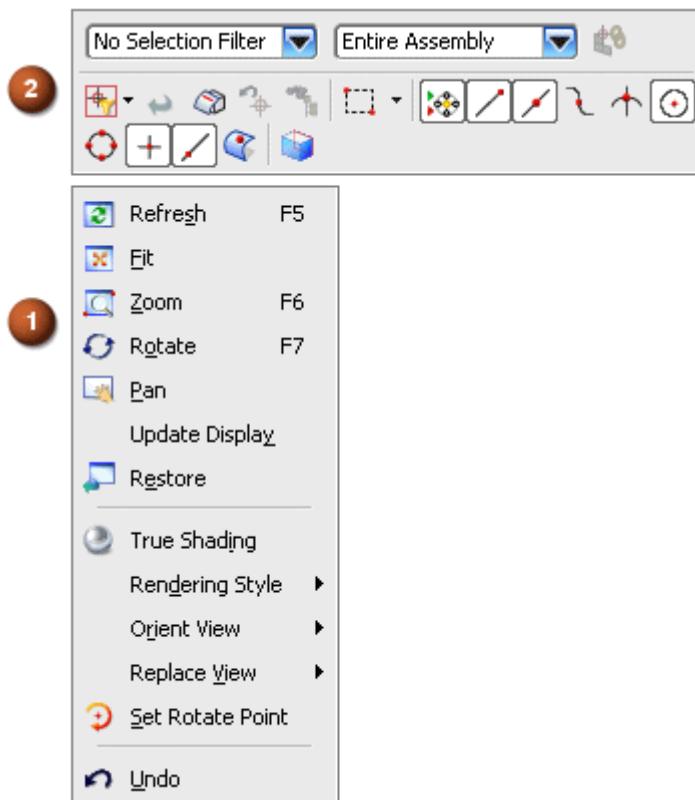
Over buttons on a toolbar	Display <i>Balloon Help</i> for the button.
Over buttons in a dialog box	Display the button name.
Over objects, features or components in graphics window	Pre-highlight objects based on the Selection Type Filter .

View shortcut menu

Right-click in the background of the graphics window to display the View shortcut menu (1). This menu lists frequently used NX commands.

 The Selection MiniBar (2) is a compact version of the Selection bar that displays in the graphics window whenever the View shortcut menu is in use. This provides convenient access to selection options close to your cursor location.

2



Option	Description
Refresh	Refreshes the entire graphics window. Erases temporary display entities.
Fit	Fits the entire part to the view. Utilizes the fit percentage found in the Preferences → Visualization → Screen dialog box.
Zoom	Fits the view to a user specified rectangle.
Rotate	Activates the rotate mode to rotate the view with the cursor.
Pan	Activates pan mode to pan the view with the cursor.
Rendering Style	Specifies the method of shading and hidden edges in which the model is displayed.
Orient View	Displays the current view in a canned view orientation. The original visualization settings and view modifications are retained. Active only in modeling view.
Set Rotate Point	Defines a point about which the model is rotated.
Clear Rotate Point	Removes a rotate point that was previously set.
Undo	Removes the effect of the last single operation performed.

Radial toolbars overview

Radial toolbars are available when you click and hold the right mouse button.

A radial toolbar with view commands appears when you click and hold the right mouse button in the background of the graphics window.

A radial toolbar with context-sensitive commands appears when you click and hold the right mouse button on an object.



Radial toolbar with view commands



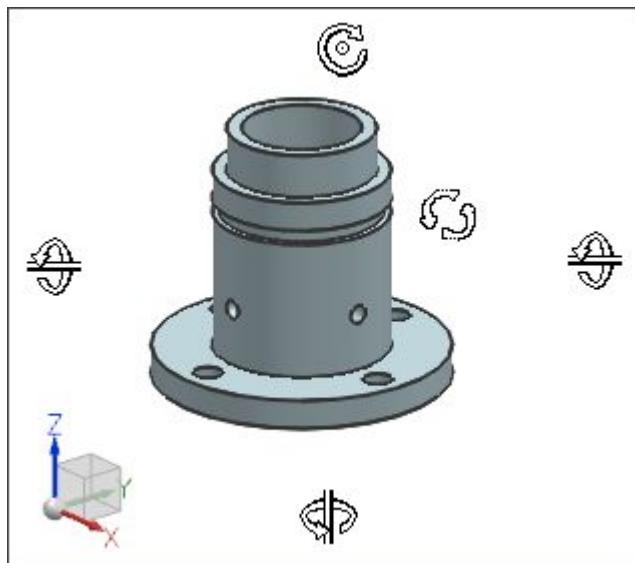
Context-sensitive radial toolbar

Graphics window view manipulation

You can rotate the view by dragging with the middle mouse button. Release the mouse button to stop rotating.

If the cursor is near the boundary of the graphics window, you can use inferred rotation about a horizontal, vertical, or normal axis.

If the cursor is in the middle of the graphics window, the axis of rotation is determined by the direction in which you drag the cursor.



Other options to manipulate the view orientation are described below.

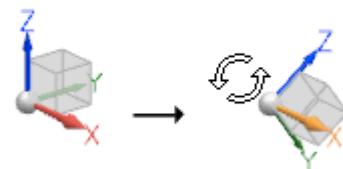
Orient View button	Modifies the orientation of a specified view to a predefined view. Changes only the alignment of the view, not the view name. This option can be invoked from the View toolbar or from the shortcut menu.
Home key	Orients the current view to Trimetric .
End key	Orients the current view to Isometric .
F8 key	Orients the current view to a selected planar face or datum plane or the planar view (top, front, right, back, bottom, left) that is <i>closest</i> to the current view orientation.

View triad

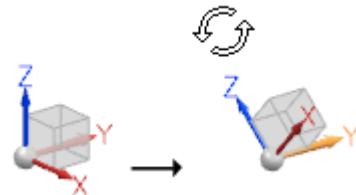
The View Triad is a visual indicator that represents the orientation of the Absolute coordinate system of the model.

- The View Triad is displayed in the lower-left corner of the graphics window.
- Select an axis of the view triad to lock rotation about that axis only.

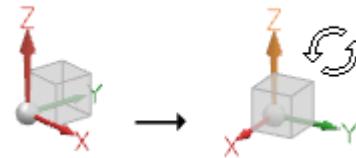
X-axis locked



Y-axis locked



Z-axis locked

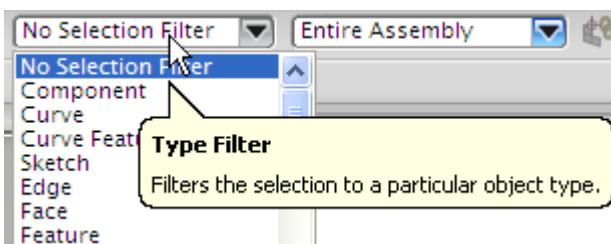


Click the middle mouse button, press Esc, or click the rotation triad origin handle to return to normal rotation.

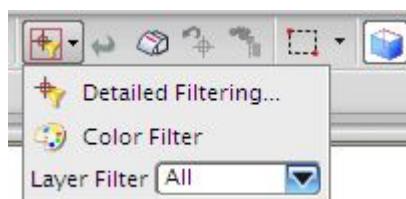
Selecting objects

You may either select an object first and then choose a command to perform, or, choose a command first and then select the required object.

The selection **Type Filter** allows you to control which type of objects you can select. The content of the list changes with the active NX command.



The **General Selection Filters** allow you to further restrict what type of objects you can select.



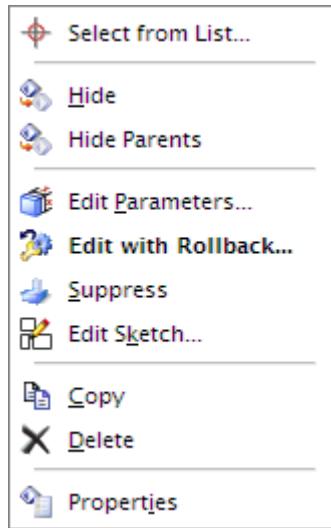
You can use toolbar options to add many additional buttons to the Selection bar.

If you right-click an object, a shortcut menu appears with commands for the object type.



The cursor must be over the object, and the object must be highlighted.

The shortcut menu changes depending on the object. The following shortcut menu is for a typical feature.



Options also vary with the application (Modeling, Drafting, Manufacturing, etc.)

Deselecting objects

You can deselect an object by holding the Shift key as you click it.

To deselect all objects in the graphics window, press the Esc (Escape) key.

Preview selection

Objects are highlighted in the preview selection color as the selection ball passes over them.

By default, **Preview Selection** is enabled. Turn it off by choosing **Preferences**→**Selection** from the menu bar.

The color of preview highlighting is determined by the **Preselection** setting found under **Preferences**→**Visualization**→**Color Settings**.

When you hold the Shift key, the preselection color is applied to currently selected objects that you can deselect.

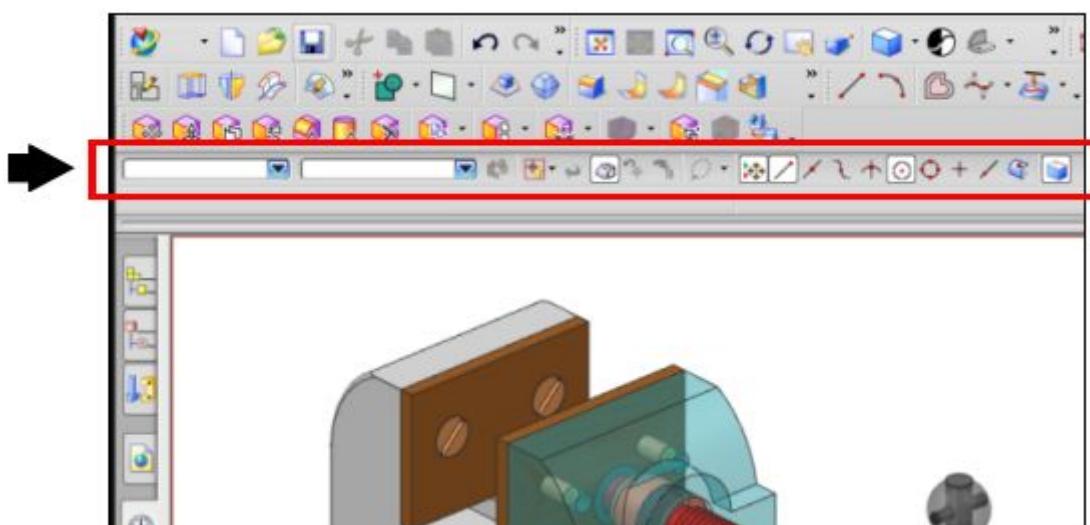
Selection bar overview

Use the *Selection bar* to set and use advanced selection options.

With the Selection bar you can:

- Select objects by filtering them for specific attributes.
- Specify how to perform multiple selections.
- Access advanced selection tools, such as Selection Intent and Snap Point.

The Selection bar appears below the toolbars found at the top of the NX window.



The main components of the Selection bar are:



- ① Selection
- ② Selection Intent
- ③ Snap Point

Where do I find it?

Toolbar	Selection bar
Menu	Edit® Selection

Selection bar Snap Point options



Button	Name	Description
	Enable Snap Point	Use this to turn the snap point options on and off.
	End Point	Select end points of lines, arcs, conics, splines, and all edge types.
	Mid Point	Select mid points of lines, open arcs, and all edge types.
	Control Point	Select a control point of a geometric object.
	Intersection Point	Select a point at the intersection of two curves.
	Arc Center	Select an arc center point.
	Quadrant Point	Select one of four quadrant points of a circle.
	Existing Point	Select an existing point.
	Point on Curve	Select a point on a curve.
	Point on Surface	Select a point on a surface.
	Tangent Point	Select a tangent point on circles, conics, and solid edges.
	Two-curve Intersection	Select the intersection point of two objects that do not fit within the selection ball by selecting two objects.
	Point Constructor	Open the Point dialog box.

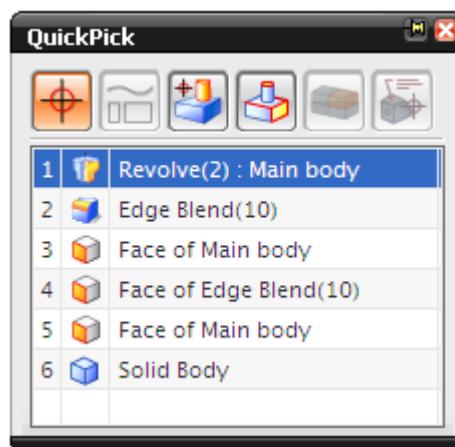
QuickPick

When you select objects, more than one object will often be within the selection ball. **QuickPick** provides easy browsing through selection candidates.

If there is more than one selectable object at the selection ball location and the cursor lingers for a short period of time, the cursor changes to a **QuickPick** indicator:



This cursor display indicates that there is more than one selectable object at that position. Click after the cursor changes to display the **QuickPick** dialog box.



You can change the amount of time the cursor must be stationary for the **QuickPick** indicator to appear.

- Choose **Preferences→Selection**.
- In the **QuickPick** group, change the **Delay** value (in seconds).



Use the middle mouse button to cycle through the items in the list and then click when the desired object is highlighted.

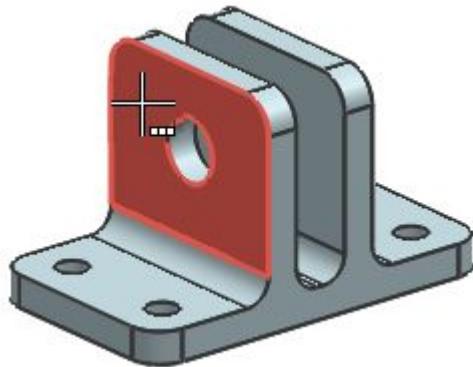
Use the buttons in the dialog box to filter the list to include object types:

- All Objects
- Construction
- Features
- Body objects
- Components
- Annotations

Activities: User Interface — Views

In the *NX user interface* section, do the activity:

- *Change the view display*



Summary: User interface

In this lesson you:

- Modified the location and contents of toolbars.
- Applied a role to restore saved toolbar settings.
- Manipulated the work view orientation.

Lesson

3 *Sketching*

3

Purpose

This lesson introduces the methods of creating a sketch.

Objectives

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

- Create a sketch.
- Create sketch curves.
- Apply dimensional constraints to sketches.
- Apply geometric constraints to sketches.
- Identify constraints.
- Convert sketch curves and constraints to reference status.

Create Sketch overview

Use the **Sketch** command when you need to:

- Define the sketch **Orientation** and **Origin**, in a manner that is not available using the quick pick method.
- Create a sketch **On Path** or associated to the edge of a solid.

3

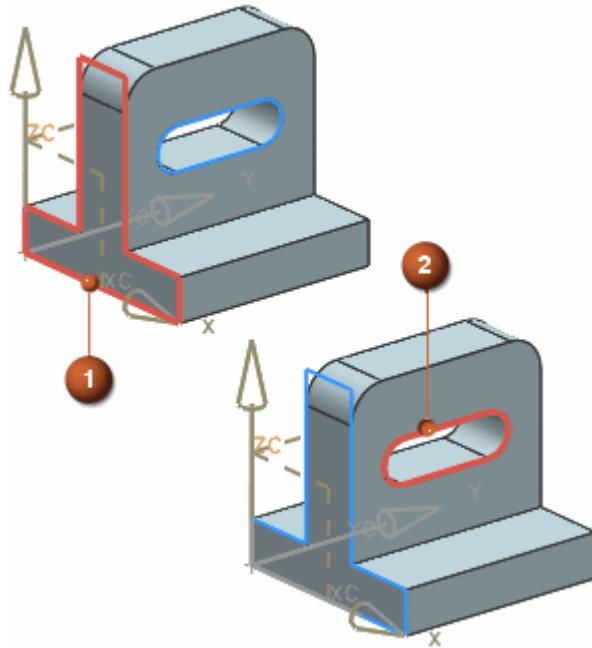
Where do I find it?

Toolbar	Direct Sketch® Sketch 
	Feature® Sketch in Task Environment 
Menu	Insert® Sketch
	Insert® Sketch in Task Environment

Sketch On Plane

Sketch On Plane

Create a **Sketch On Plane** when you want to associate the sketch feature to a planar object such as a datum plane or a face.



3

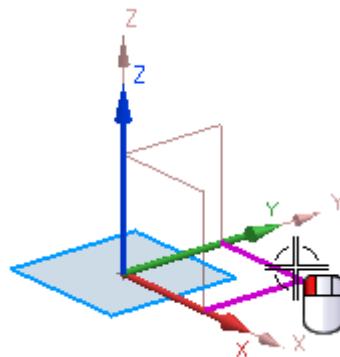
1. Sketch on the plane of a Datum CSYS
2. Sketch on a face of the extruded sketch.

Create a sketch On Plane

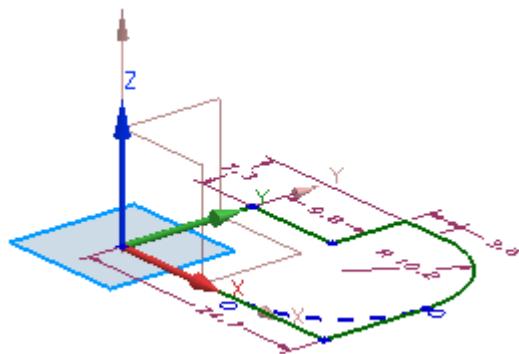
Use this procedure to sketch on a plane.

1. On the **Direct Sketch** toolbar, click **Profile** .

2. Define the sketch plane by selecting an existing plane.

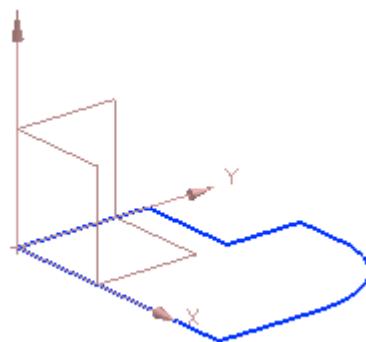


3. Create sketch curves.



4. Define needed sketch constraints or dimensions.

5. Choose another command or click **Finish**, on the **Direct Sketch** toolbar.



Create a sketch On Face

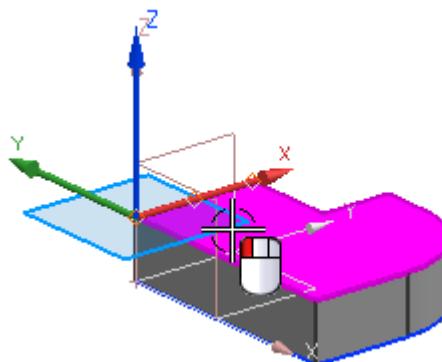
Use this procedure to sketch on a face when your design intent requires the sketch to move when the faces moves.

1. On the **Direct Sketch** toolbar, click **Profile** 
2. Define the sketch plane by selecting an existing face.

The first pick defines the sketch:

- Plane
- Orientation
- Origin
- First point for the profile.

3



You can hold Alt and click the face to define the plane without defining the first point.

3. Create sketch curves.
4. Define needed sketch constraints or dimensions.
5. Choose another command or click **Finish**, on the **Direct Sketch** toolbar.

Direct Sketch and the Sketch task environment

The **Direct Sketch** toolbar and the **Sketch task environment** offer two modes you can use to create sketches.

Use the **Direct Sketch** toolbar when you want to:

- Create a new sketch in the current 3D orientation.
- See the effect of sketch changes on the model in real-time.
- Edit a sketch that has a limited number of down-stream features.

Use the **Sketch task environment** when you want to:

- Create a new sketch in a 2D orientation.
- Edit an internal sketch.
- Edit a sketch that has a significant number of down-stream features.
- Access additional commands like **Project Curve**, **Intersection Point**, and **Intersection Curve**.
- Experiment with sketch changes, but retain the option to discard the changes.

Direct sketching

A **Direct Sketch** toolbar is available in Modeling. Use the commands on this toolbar to create a sketch on a plane without entering the Sketch task environment.

When you create a point or curve using the commands on this toolbar, a sketch is created and is active. The new sketch is listed in the model history in the **Part Navigator**. The first point you specify defines the sketch plane, orientation, and origin.

You can define the first point on the following:

- Screen position
- Point
- Curve
- Face
- Plane
- Edge
- Specified Plane
- Specified Datum CSYS

3

Why should I use it?

Direct sketching requires fewer mouse clicks, which makes creating and editing sketches faster and easier.

Where do I find it?

Application	Modeling
Toolbar	Direct Sketch



Sketch Task Environment overview

The Sketch Task Environment command lets you create or edit a sketch in NX while allowing full control over the creation and edit process.

The Sketch Task Environment is similar to a separate application, where the interface changes to focus on the current toolset, but you are still in the Modeling application. All the toolbars shown support the sketch tool.

For many features where internal sketching is supported, the Sketch Task Environment is the method used to create or edit the sketch.

Working in the Sketch Task Environment allows the following:

- The ability to control the sketch creation options during sketch creation.
- Access to all the sketch tools.
- The option to work in 2D or 3D space. (The default is orient to 2D.)
- The ability to control the update behavior of the model.

Where do I find it?

Application	Modeling
Toolbar	Feature ® Sketch in Task Environment
Menu	Insert ® Sketch in Task Environment

Sketches and Layers

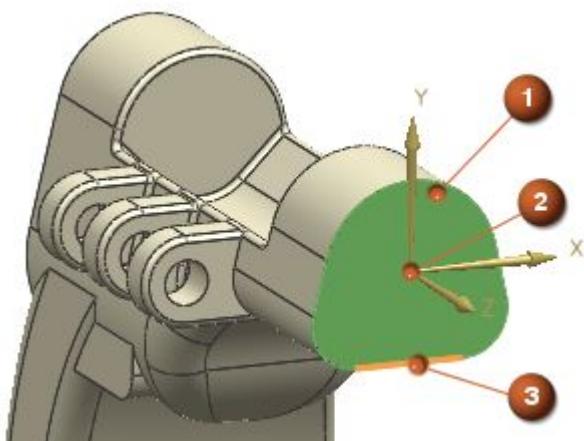
Layers can be used as an organization tool for sketches.

- You can **Hide** and **Show** sketches from the **Part Navigator**. You do not need to put each sketch on a different layer to control its visibility.
- While working in an external sketch, all the objects you create reside on the same layer.
 -  Internal sketches reside with the parent feature unless you manually move the sketch to another layer.
- When you open a sketch, the layer that the sketch resides on becomes the work layer.
- When you add curves to an active sketch, they are automatically moved to the same layer as the sketch.
- When you exit a sketch, the layer settings depend on the whether you selected the **Maintain Layer Status** check box in the **Preferences® Sketch® Session Settings** dialog box.
 - If you select the check box, the sketch layer and work layer are returned to the status they had before you activated the sketch.
 - If you clear the check box, the sketch layer continues to be the work layer.

Sketch reference direction

When you choose a reference direction, vertical or horizontal, you determine either the positive X or Y direction for your sketch.

NX is a counterclockwise system. Your sketch final Z axis orientation should point towards you in the graphics window. This ensures that your sketch will respond according to positive feature creation assumptions and the right hand rule.



1. Sketch plane
2. Sketch Origin and CSYS. The final orientation of the sketch is based on this preview CSYS.
3. Sketch Orientation. The horizontal reference direction is used to properly orient the sketch.

Intermediate Datum CSYS

NX can automatically create an **Intermediate Datum CSYS** when you create or reattach a sketch. By default, the **Intermediate Datum CSYS** is internal to the sketch. It is not visible outside the sketch or in the model history in the **Part Navigator**.

Why should I use it?

Select this option to associate the **Intermediate Datum CSYS** to the base feature used to create the sketch. This option also gives the sketch independence so that if you delete the base feature the sketch remains.

You can make this Datum CSYS:

- External from the sketch when you right-click the sketch in the **Part Navigator** and choose **Make Datums External**.
- Internal to the sketch when you right-click the sketch in the **Part Navigator** and choose **Make Datums Internal**.

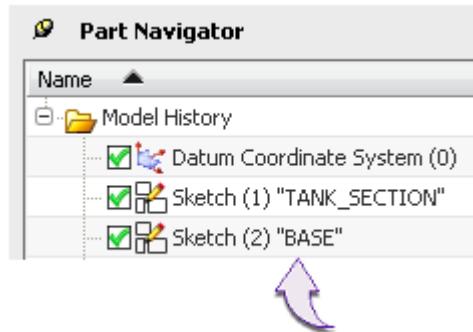
Where do I find it?

Toolbar	(Modeling) Direct Sketch® Sketch  (Sketch task environment) Feature® Sketch in Task Environment 
Menu	(Modeling) Insert® Sketch (Sketch task environment) Insert® Sketch in Task Environment
Location in dialog box	Sketch Plane group® Inferred Settings group® Create Intermediate Datum CSYS

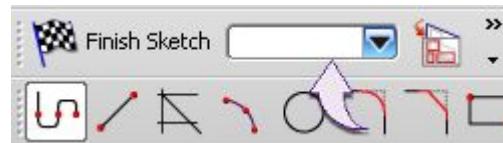
Name sketches in Sketch Task Environment

Sketches are assigned a default name with a numeric suffix such as Sketch (1) "SKETCH_000".

Once you name your sketch the new name displays in the Part Navigator, for example, Sketch (2) "BASE".



1. On the **Sketch** toolbar, clear the contents of the Sketch Name box.



2. Type a new name and press Enter.



It is recommended that you name your sketch when no other sketch commands are active.

If your new sketch name *does not* appear in the **Part Navigator**, make sure **Timestamp Order** is in effect.



Finish Sketch

Use the **Finish Sketch** command to exit a sketch and return to the application or command you started sketching from.

Where do I find it?

	(Modeling) Direct Sketch® Finish Sketch
Toolbar	(Sketch task environment) Sketch Tools® Finish Sketch
Menu	(Modeling) File® Finish Sketch (Sketch task environment) Task® Finish Sketch
Keyboard	Ctrl+Q

Exit Sketch

Use the **Exit Sketch** command to exit the Sketch task environment without doing the following:

- Saving the modified sketch.
- Updating the model.

The **Exit Sketch** command restores the part to the state it was prior to entering the Sketch task environment while avoiding the unnecessary step of updating all features downstream of the sketch.

Why should I use it?

Use this command when you want to do the following:

- Explore and edit sketch curves and constraints without changing sketches and the model permanently.
- Discard sketch edits made since you entered the Sketch task environment.

Where do I find it?

Task environment	Sketch
Menu	Task® Exit Sketch

Direct sketch and feature edit preferences

Set the following in the **Modeling Preferences** dialog box for the actions available when you double-click sketches and features.

Edit tab	
Edit Sketch Action	Direct Edit Lets you edit the sketch directly in Modeling.  Direct Edit updates the model immediately. Select Task Environment if you have sketches with many dependent features.
Double-click Action (Sketches)	Task Environment Enters the Sketch task environment. Edit with Rollback Makes the selected sketch the current feature and enters the Sketch task environment.
Double-click Action (Features)	Edit Enters the direct sketch mode without changing the current feature. Edit with Rollback Makes the selected feature the current feature and enters the edit mode.
	Edit Enters the feature edit mode without changing the current feature.

To edit sketches directly in Modeling, you should:

- Set **Double-click Action (Sketches)** to **Edit with Rollback**.
- Set **Edit Sketch Action** to **Direct Edit**.
- Right click the sketch and choose  **Edit**.

Where do I find it?

Application	Modeling
Menu	Preferences® Modeling
Location in dialog box	Edit tab

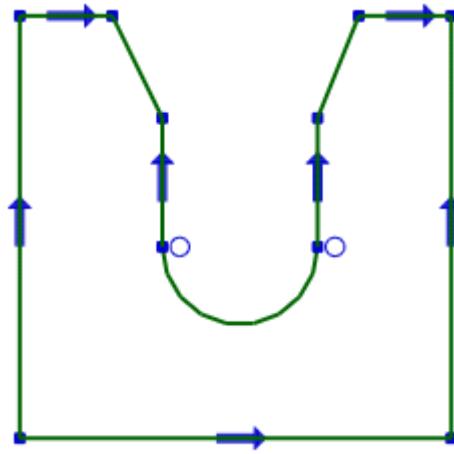


Profile overview

Use the **Profile** command to create a series of connected lines and/or arcs in string mode.

In string mode, the end of the last curve becomes the beginning of the next curve.

For example, you can create this pipe vise profile in one series of mouse clicks:

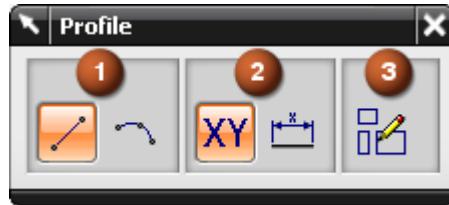


3

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Profile  (Drafting and Sketch task environment) Sketch Tools® Profile 
Menu	(Modeling and Drafting) Insert® Sketch Curve® Profile (Sketch task environment) Insert® Curve® Profile

Profile dialog bar



3

1) Object Type

Line



Creates a line.

This is the default mode when you choose **Profile**.

Points selected off of the sketch plane are projected onto the sketch plane.

Arc



Creates an arc.

While stringing from line to arc, you create a two point arc. You can create a three point arc if the first object drawn in string mode is an arc.

By default, **Profile** switches to line mode after you create an arc. To create a series of chained arcs, double-click the **Arc** option.

2) Input Mode

Coordinate Mode



Creates curve points using X and Y coordinate values.

Parameter Mode



Creates curve points using parameters that are appropriate to a line or arc curve type.

Lines: **Length** and **Angle** parameters.

Arcs: **Radius** and **Sweep Angle** parameters.

3) Sketch

Define Sketch Plane



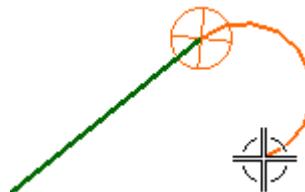
Lets you define a sketch plane using standard plane creation methods.

Create Profile sketch curves

This is the general process for creating a series of connected lines and/or arcs in the string mode using the command, **Profile** 

1. In the graphics window, click once to establish a start point.
2. Move to the end of the desired line, and click once to establish the end point.
3. To create an arc, click and drag anywhere in the graphics window to switch from line creation to arc creation.
4. Move the cursor away from the line end through different quadrants of the circle, without clicking, to establish the direction of the arc.

3



Perpendicular or tangential relationships change depending on the quadrant through which you move away from the line end.

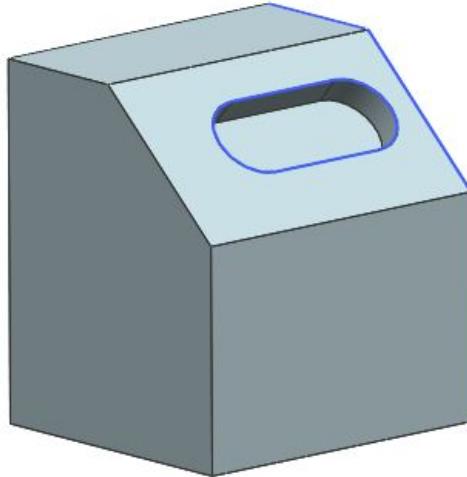
5. Click once to establish the end of the arc.
6. Line mode is automatically restored after an arc is created. To create consecutive arcs, double-click the **Arc** button any time during profile mode.
7. To stop the profile string mode, click the middle mouse button.

Activities: Sketching

In the *Sketching* section, do the activities:

- *Sketch on a planar face*

3



Sketch curves

Sketching commands are available on both the Direct sketching toolbar and in the Sketch Task Environment.

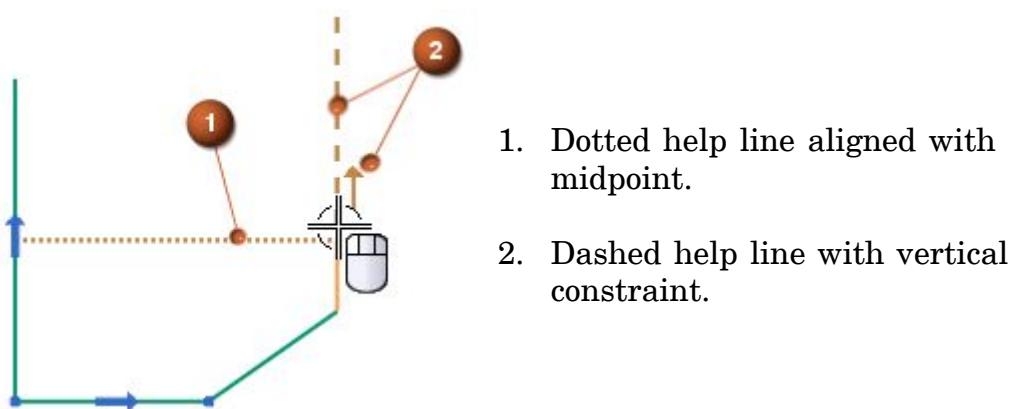
The following is a chart of the essential sketching commands. Not all available commands are shown.

	Profile	Creates a series of connected lines and/or arcs in string mode; that is the end of the last curve becomes the beginning of the next curve.
	Line	Creates lines with constraint inferences.
	Arc	Creates an arc through three points or by specifying its center and end points.
	Circle	Creates a circle through three points or by specifying its center and diameter.
	Rectangle	Creates a rectangle using any of three different methods.
	Studio Spline	Dynamically creates and edits splines by dragging defining points or poles, and assigning slope or curvature constraints at defining points.
	Point	Creates points.
	Offset Curve	Offsets a chain of curves, projected curves, or curves/edges in the sketch. Symmetric offset is also available.

Sketch help lines

Help lines indicate alignment to control points of curves, including line endpoints and midpoints, arc endpoints, and arc and circle centerpoints. Two types of help lines can display during curve creation:

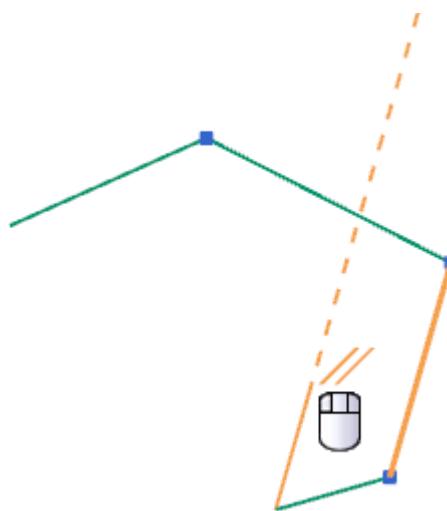
- A dotted help line indicates alignment with other objects.
- A dashed help line indicates an inferred constraint with other objects, such as horizontal, vertical, perpendicular and tangent.



Short List

NX maintains a memory of five objects called the short list to check for inferred constraints. NX uses the short list to avoid testing against every curve in the current sketch when inferring constraints. Curves are added to the list as you create them or when you pass your cursor over a curve.

When NX infers a constraint between an object on the short list and the curve being created, the short list object highlights and you see a preview of the constraint.



- The short list is structured from top to bottom. When curves are created or passed over with the cursor, they are placed on top of the short list. When the list is full, the curve at the bottom is removed before adding a new curve at the top.
- NX clears the short list when you leave a curve command.



Continuous Auto Dimensioning

Use the **Continuous Auto Dimensioning** command to automatically dimension sketch curves after each operation.

This command uses auto dimensioning rules to fully constrain the active sketch, including positioning dimensions to the parent Datum CSYS.

You can set auto dimensioning rules from the following:

- **Inferred Constraints and Dimensions**
- **File® Utilities® Customer Defaults® Sketch→Inferred Constraints and Dimensions® Dimensions**

The **Continuous Auto Dimensioning** command creates the automatic dimension type of sketch dimension.

Automatic dimensions fully constrain a sketch. The dimensions are updated as you drag the sketch curves. They remove degrees of freedom from the sketch but do not lock the values permanently. If you add a constraint that conflicts with an automatic dimension, the automatic dimension is deleted. You can convert automatic dimensions into driving dimensions.

Why should I use it?

In Modeling, use this command to ensure you are always working with a fully constrained sketch which will be updated predictably.

In Drafting, use this command to automatically create dimensions for all the curves that you create in a drawing.

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Continuous Auto Dimensioning  / Inferred Constraints and Dimensions  (Drafting and Sketch task environment) Sketch Tools® Continuous Auto Dimensioning  / Inferred Constraints and Dimensions 
Menu	(Modeling, Drafting, and Sketch task environment) Tools® Sketch Constraints® Continuous Auto Dimensioning / Inferred Constraints and Dimensions

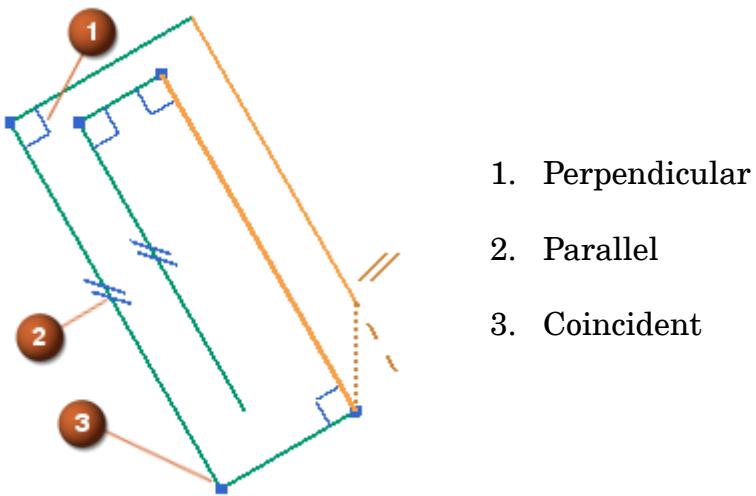


Inferred Constraints and Dimensions overview

Use the **Inferred Constraints and Dimensions** command to control which constraints or dimensions are automatically inferred during curve construction.

You can set inferred constraints for geometric constraints, dimensional constraints, and constraints recognized when using snap point options.

3



1. Perpendicular
2. Parallel
3. Coincident



Inferred constraints behave like normally applied geometric constraints and can be seen and deleted using the **Show/Remove Constraints** dialog or when you right-click an object and choose **Remove All Constraints**.

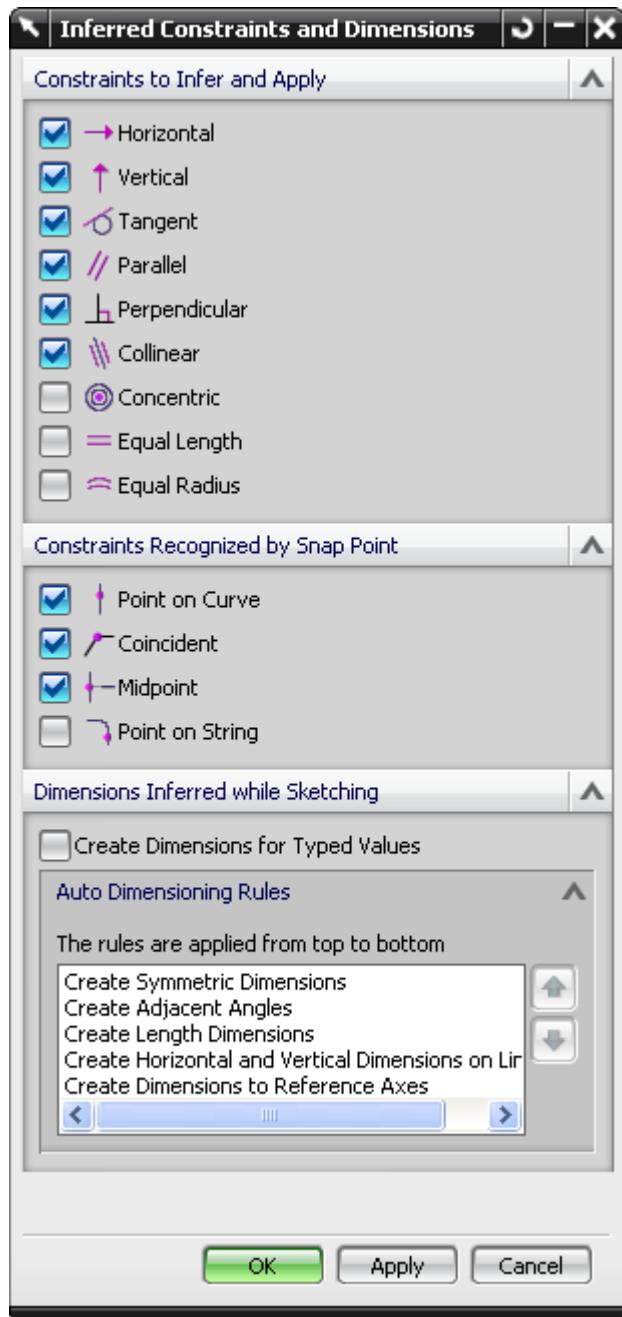


You can temporarily disable all of the infer constraint settings during curve construction by pressing and holding the Alt key on Windows or the Ctrl+Alt keys on non-Windows platforms.

Where do I find it?

Task environment	Sketch
Toolbar	Sketch Tools → Inferred Constraints
Menu	Tools → Constraints → Inferred Constraints

Inferred Constraints and Dimensions dialog box



Available constraints

Constraints recognized by Snap Point

Auto Dimensioning rules

Inferred Constraints and Dimensions options

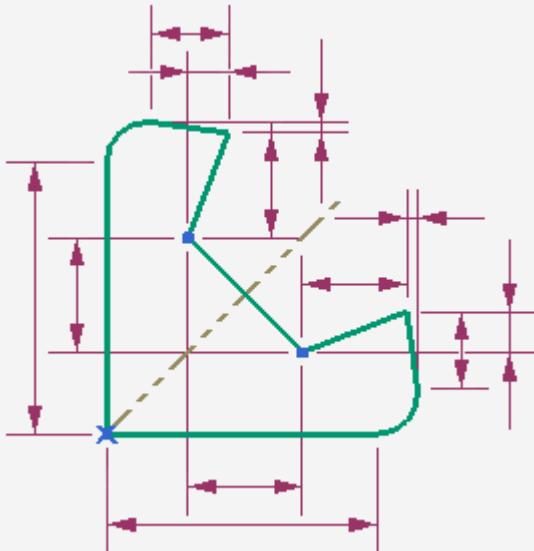
Dimensions Inferred while Sketching

Create Dimensions for Typed Values Creates a driving dimension for any geometry created by typing a value in an on-screen input box.

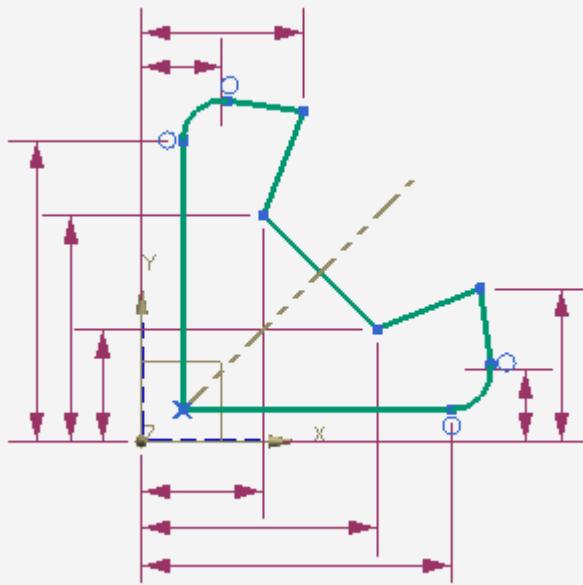
Auto Dimensioning Rules

The following are the rules for creating automatic dimensions. You can apply these rules in any order.

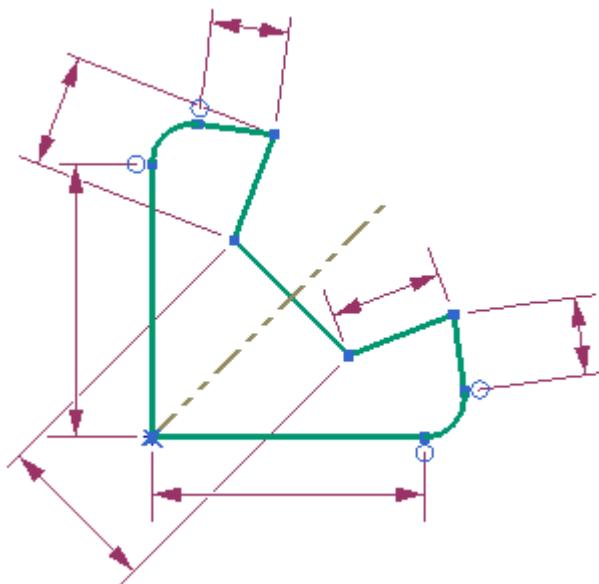
Create Horizontal and Vertical Dimensions on Lines



Create Dimensions to Reference Axes

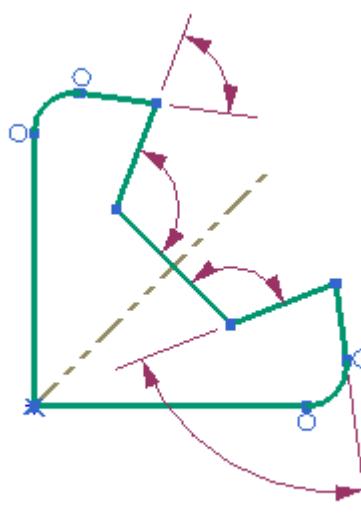


Create Length Dimensions



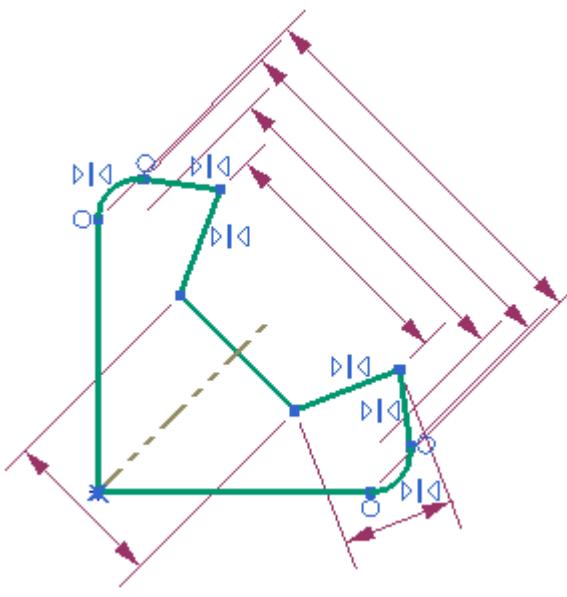
Create Adjacent Angles

Creates angular dimensions between adjacent objects.



Create Symmetric Dimensions

Creates symmetric dimensions if there are any symmetric constraints or mirrored curves. The dimensions will be created between these objects across the symmetry line



The Snap Angle option

You can specify the snap angle on the Session Settings page of the **Sketch Preferences** dialog box.

The **Snap Angle** option lets you specify the value of the default snap angle tolerance for vertical, horizontal, parallel, and perpendicular lines.

The default snap angle is 3 degrees.

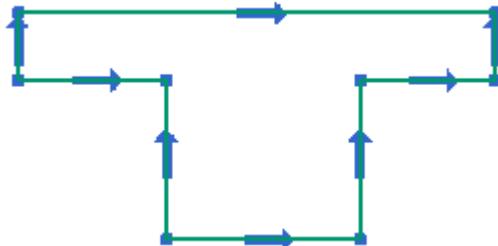
The snap angle must be greater than zero and less than 20 degrees.

Hold the Alt key to temporarily disable the snap action.

Activities: Sketching curves

In the *Sketching* section, do the activities:

- *Create a sketch profile*



Sketch curve functions

There are several options you can use to modify sketch curves:

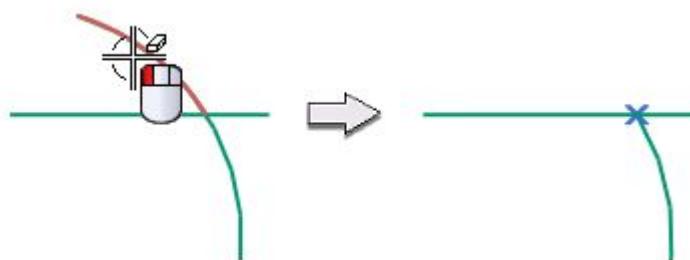
Quick Trim		Trims curves to the closest physical or virtual intersection.
Quick Extend		Extends curves to a physical or virtual intersection.
Make Corner		Creates a corner by extending and/or trimming two input curves to a common intersection.
Fillet		Creates a fillet between two or three curves.
Chamfer		Bevels a sharp corner between two sketch lines.



Quick Trim overview

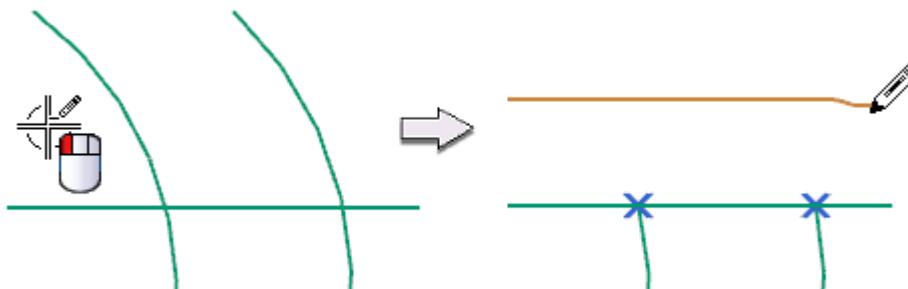
Use the **Quick Trim** command to trim a curve to the closest physical or virtual intersection in either direction. You can:

- Preview the trim by passing the cursor over the curve.
- Select individual curves to trim.



3

- Hold the left mouse button and drag across multiple curves to trim them all at the same time.



Trimming a curve that has no intersection deletes the curve.

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Quick Trim  (Drafting and Sketch task environment) Sketch Tools® Quick Trim 
Menu	(Modeling and Drafting) Edit® Sketch Curve® Quick Trim (Sketch task environment) Edit® Curve® Quick Trim

Sketching constraints and Quick Trim

When the **Create Inferred Constraints**  command is used, the appropriate constraints are inferred after a trim operation.

Constraint	Trim Operation
Concentric	An arc in the middle.
Coincident	Intersecting lines to an intersection point.
Point on Curve	A curve to a boundary curve.
Collinear	A line in the middle
Equal Radius	An arc in the middle.
Tangent	A curve at the tangent point of a boundary curve.

Use Quick Trim

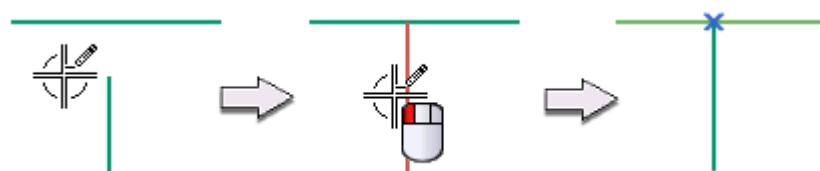
1. On the **Direct Sketching** toolbar, click **Quick Trim** .
2. For the **Curve to Trim**, select either a single curve or use the drag path method to select multiple curves.
 The Curve to Trim will trim to the closest intersecting curve. If you want to trim to a different curve select the **Boundary Curve** first.
3. Conditional: In the **Settings** group, select the **Trim to Extension** box if you selected a boundary curve that will produce a theoretical intersection with the Curve to Trim.



Quick Extend overview

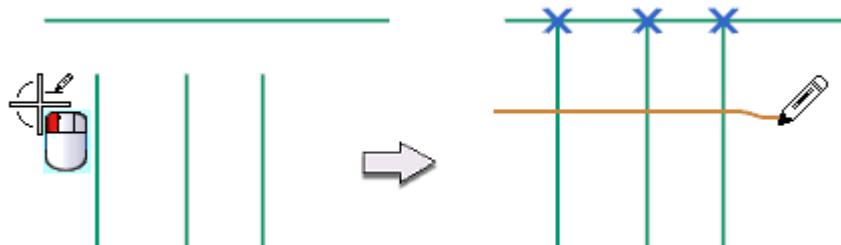
Use the **Quick Extend** command to extend a curve to a physical or virtual intersection with another curve. You can:

- Preview the extension by passing the cursor over the curve.
- Select individual curves to extend.



3

- Hold the left mouse button and drag across multiple curves to extend them all at the same time.



Where do I find it?

Toolbar	(Modeling) Direct Sketch® Quick Extend  (Drafting and Sketch task environment) Sketch Tools® Quick Extend 
Menu	(Modeling and Drafting) Edit® Sketch Curve® Quick Extend (Sketch task environment) Edit® Curve® Quick Extend

Sketcher constraints and Quick Extend

When the **Create Inferred Constraints**  command is used, the appropriate constraints after an extend operation are inferred.

Constraint	Extend Operation
Coincident	A curve to the endpoint of another curve.
Point on Curve	A curve to a boundary curve.
Tangent	A curve to a tangent point of a boundary curve.

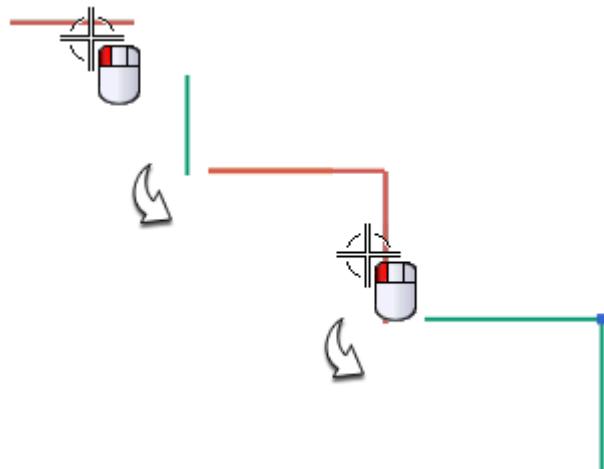
Use Quick Extend

1. On the **Direct Sketch** toolbar, click **Quick Extend** .
2. For the **Curve to Extend**, select either a single curve or use the drag path method to select multiple curves.
 The Curve to Extend will extend to the closest intersecting curve. If you want to extend to a different curve select the **Boundary Curve** first.
3. Conditional: In the **Settings** group, select the **Extend to Extension** box if you selected a boundary curve that will produce a theoretical intersection with the Curve to Extend.



Make Corner overview

Use the **Make Corner** command to create a corner by extending and/or trimming two input curves to a common intersection. If the **Create Inferred Constraints** option is on, a coincident constraint is created at the intersection.



3

Make Corner works with:

- Lines
- Arcs
- Open conics
- Open splines (trimming only)

You can also hold the left mouse button and drag over curves create a corner.

Where do I find it?

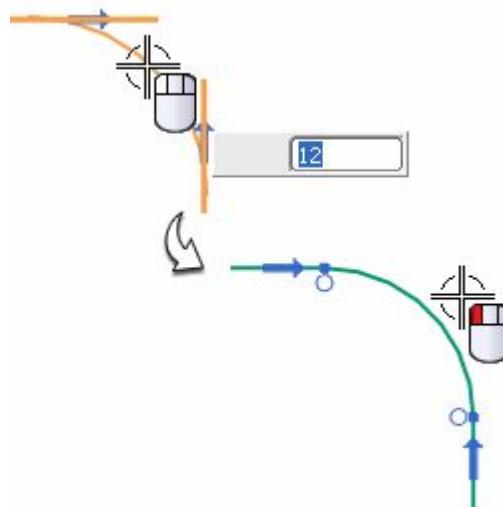
Toolbar	(Modeling) Direct Sketch® Make Corner  (Drafting and Sketch task environment) Sketch Tools® Make Corner 
Menu	(Modeling and Drafting) Edit® Sketch Curve® Make Corner (Sketch task environment) Edit® Curve® Make Corner



Fillet overview

Use the **Fillet** command to create a fillet between two or three curves. You can:

- Trim all input curves or leave them untrimmed.
- Delete the third curve of a three-curve fillet.
- Specify a value for the fillet radius, or preview the fillet and determine its size and location by moving the cursor.
- Hold the left mouse button and drag over curves to create a fillet.



Fillet preview and final output

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Fillet  (Drafting and Sketch task environment) Sketch Tools® Fillet 
Menu	(Modeling and Drafting) Insert® Sketch Curve® Fillet (Sketch task environment) Insert® Curve® Fillet



Chamfer

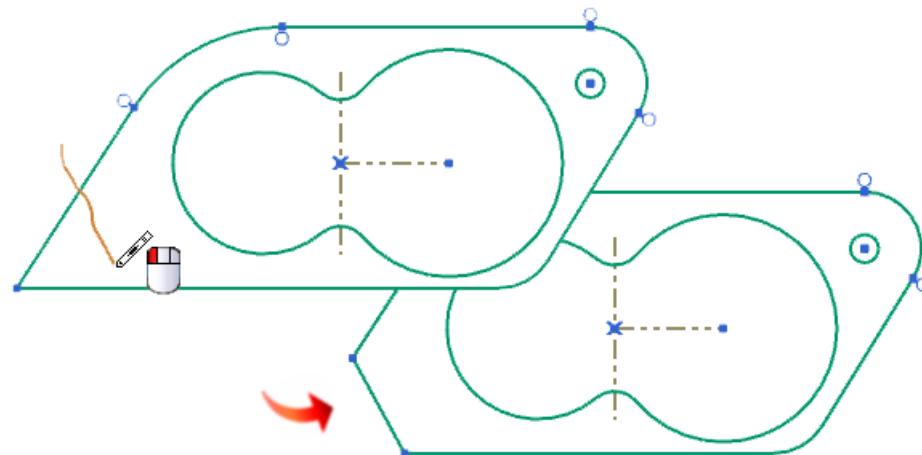
Use the **Chamfer** command to bevel a sharp corner between two sketch lines.

You can create the following chamfer types:

- Symmetric
- Asymmetric
- Offset and Angle

3

You can also hold the left mouse button and drag over curves to create a chamfer.



Where do I find it?

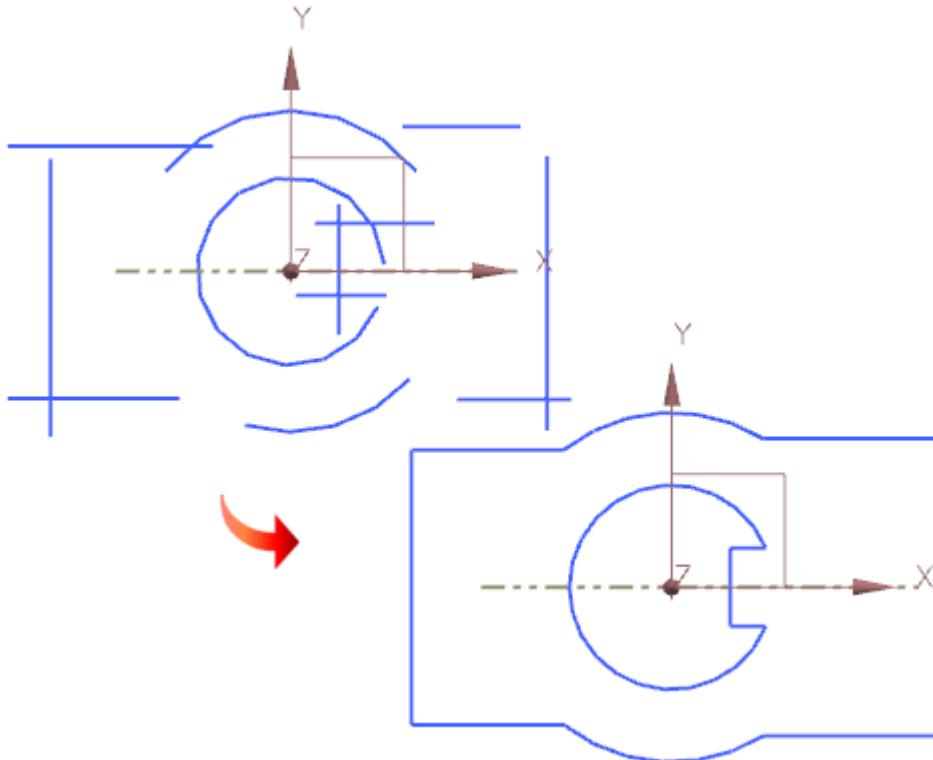
Toolbar	(Modeling) Direct Sketch® Chamfer  (Drafting and Sketch task environment) Sketch Tools® Chamfer 
Menu	(Modeling and Drafting) Insert® Sketch Curve® Chamfer (Sketch task environment) Insert® Curve® Chamfer

Activities: Quick Trim, Quick Extend, Make Corner

In the *Sketching* section, do the activities:

- *Trim and extend curves, and make corners*

3



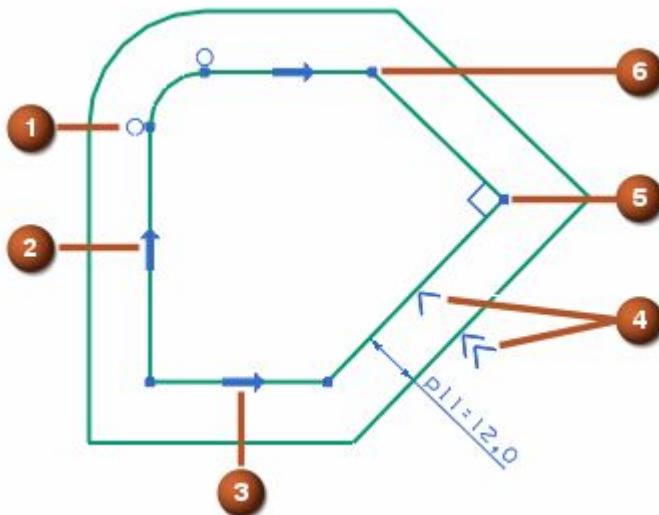
Types of constraints

Use constraints to precisely control the objects in a sketch and to express the design intent for a feature. There are two types of constraints: geometric constraints and dimensional constraints.



Geometric Constraints

A geometric constraint establishes a relationship between two or more sketch objects, for example, requiring that two lines be perpendicular or parallel, or that several arcs have the same radius.



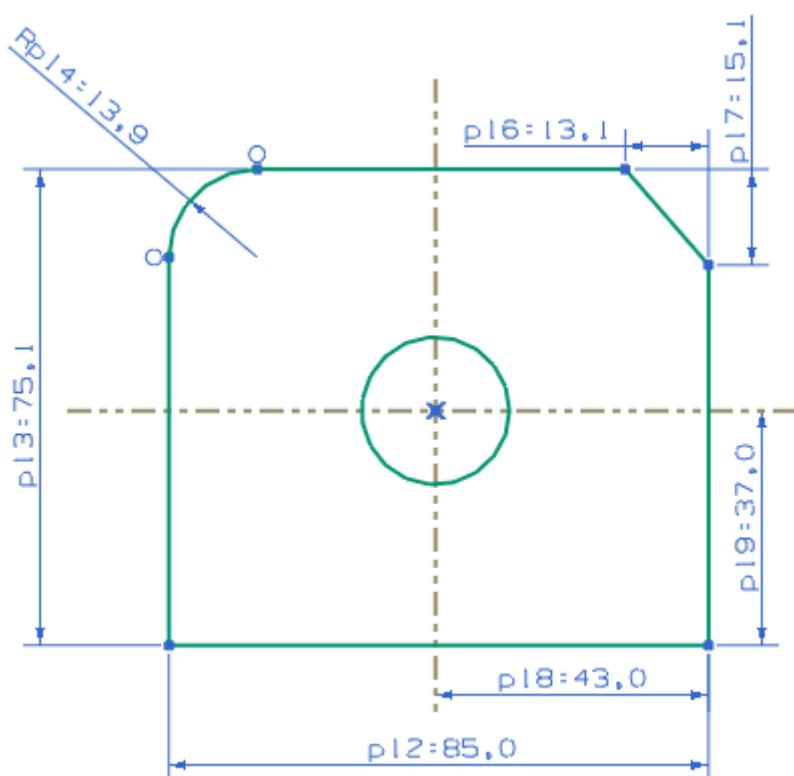
1. Tangent
2. Vertical
3. Horizontal
4. Offset
5. Perpendicular
6. Coincident



Dimensional Constraints

Dimensional constraints, also called driving dimensions, establish

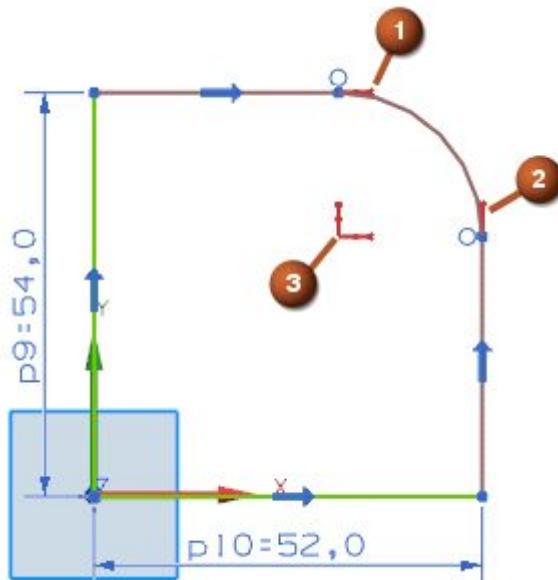
- The size of a sketch object such as the radius of an arc or length of a curve.
- A relationship between two objects, such as the distance between two points.



Dimensional constraints display like drafting dimensions in that they have dimension text, extension lines, and arrows. However, dimensional constraints dictate the size of sketch objects.

Degree-of-freedom arrows

Degree-of-freedom (DOF) arrows  mark points on a sketch that are free to move. There are three types of degree-of-freedom: positional, rotational, and radial. This example shows positional constraints:



1. This point is free to move in the X direction only.
2. This point is free to move in the Y direction only.
3. This point is free to move in both the X and Y directions.

3

When you constrain a point from moving in a given direction, NX removes the DOF arrow. Applying one constraint can remove several DOF arrows. When all the arrows are gone, the sketch is fully constrained.

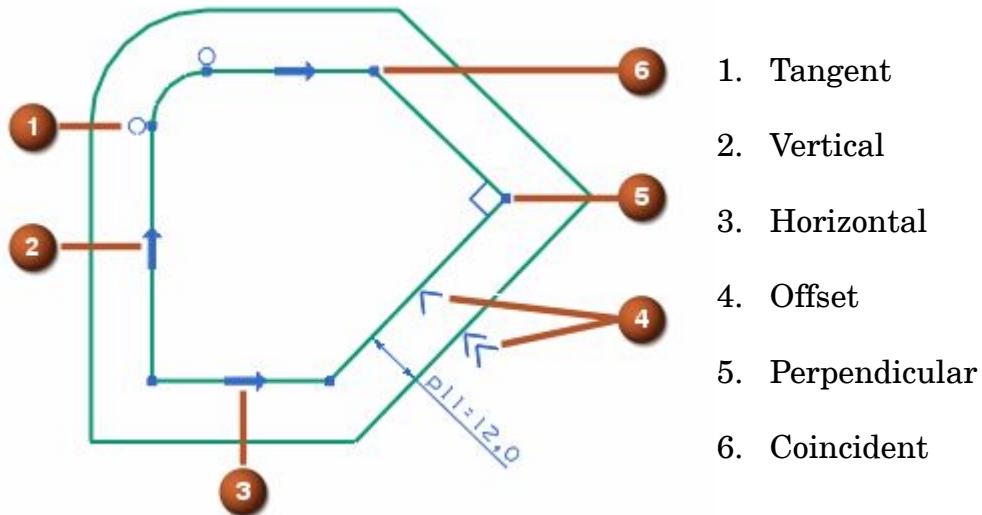
Fully constrain a sketch when you need complete control of the design. Note that constraining a sketch is optional. You can use an under-constrained sketch to define a feature.

Geometric Constraints overview

Use the **Constraints** command to add geometric constraints to sketch geometry. These specify and maintain conditions for or between sketch geometry.

Constraints can:

- Define a line as being horizontal or vertical.
- Ensure that multiple lines remain parallel to each other.
- Require that several arcs have the same radius.
- Position your sketch in space or relative to outside objects.



Where do I find it?

Toolbar	(Modeling) Direct Sketch® Constraints  (Sketch task environment) Sketch Tools® Constraints 
Menu	(Modeling) Insert® Sketch Constraint® Constraints (Sketch task environment) Insert® Constraints

Create geometric Constraints

1. On the **Sketch Tools** toolbar, click **Constraints** .
2. In the graphics window, select the sketch objects you want to constrain.
 -  You can create multiple geometric constraints without reentering the command by holding the Ctrl key while selecting the objects to be constraints.
3. On the **Constraints** dialog bar, click one of the constraint options or right-click the selected sketch objects and choose the constraint type from the list.
4. Click **Constraints** or the middle mouse button to exit the command.

3

Geometric constraints quick reference

The following table shows the essential geometric constraints. Not all possible constraints are shown.

Constraint Type	Command icon	Icon in graphics window	Description
Fixed			<p>Defines fixed characteristics for geometry, depending on the type of geometry as follows:</p> <ul style="list-style-type: none"> Point Fixes the location. Line Fixes the angle. Line, Arc or elliptical arc endpoint Fixes the location of the endpoint. Arc center, elliptical arc center, circle center, or ellipse center Fixes the location of the center. Arc or circle Fixes the radius and the location of the center. Elliptical arc or Ellipse Fixes the radii and the location of the center. Spline control point Fixes the location of the control point.
Fully Fixed			Creates sufficient fixed constraints to completely define the position and orientation of sketch geometry in one step.
Coincident			Defines two or more points as having the same location.
Concentric			Defines two or more circular and elliptical arcs as having the same center
Collinear			Defines two or more lines as lying on or passing through the same straight line.

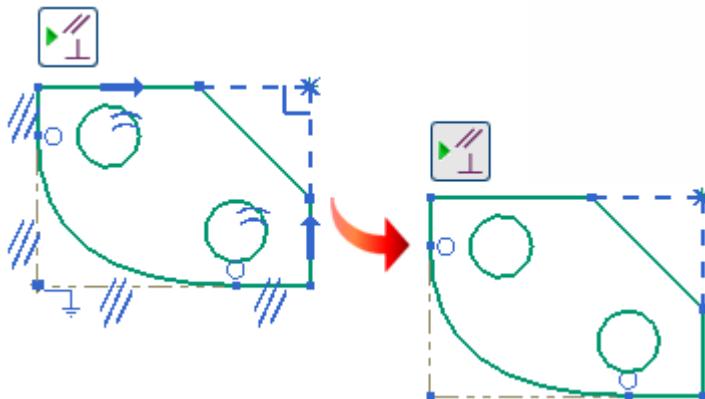
Point On Curve			Defines the location of a point as lying on a curve.
Point on String			Defines the location of a point as lying on a projected curve. You must select the point first, then select the curve. This is the only valid constraint that should be applied to a projected curve.
Midpoint			Defines the location of a point as equidistant from the two end points of a line or a circular arc. For the Midpoint constraint, select the curve anywhere other than at its endpoints.
Horizontal			Defines a line as horizontal.
Vertical			Defines a line as vertical.
Parallel			Defines two or more lines or ellipses as being parallel to each other.
Perpendicular			Defines two lines or ellipses as being perpendicular to each other.
Tangent			Defines two objects as being tangent to each other.
Equal Length			Defines two or more lines as having the same length.
Equal Radius			Defines two or more arcs as having the same radius.
Constant Length			Defines a line as having a constant length.
Constant Angle			Defines a line as having a constant angle.
Mirror Curve			Defines two objects as being mirror images of each other.
Make Symmetric			Defines two objects as being symmetric with each other.

Offset Curve			The Offset Curve command offsets a chain of curves, projected curves, or curves/edges in the current assembly, and constrains the geometry using an Offset constraint.
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Show All Constraints overview

Use the **Show All Constraints** command to show all geometric constraints applied to the sketch.



Where do I find it?

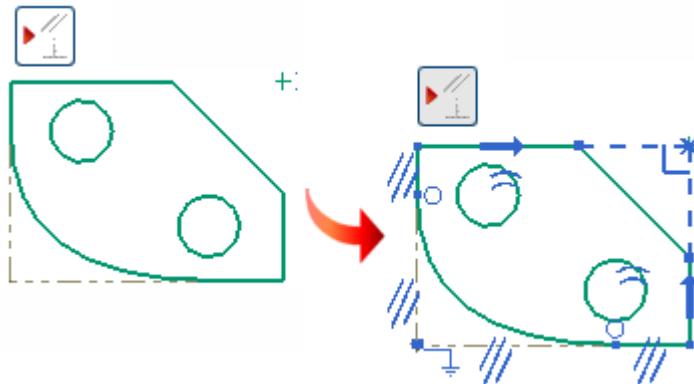
Toolbar	(Modeling) Direct Sketch® Show All Constraints (Sketch task environment) Sketch Tools® Show All Constraints
Menu	(Modeling) Tools® Sketch Constraints® Show All Constraints (Sketch task environment) Tools® Constraints® Show All Constraints

3



Show No Constraints overview

Use the **Show No Constraints** command to hide all geometric constraints applied to the sketch.



Where do I find it?

Toolbar	(Modeling) Direct Sketch® Show No Constraints (Sketch task environment) Sketch Tools® Show No Constraints
Menu	(Modeling) Tools® Sketch Constraints® Show No Constraints (Sketch task environment) Tools® Constraints® Show No Constraints



Show / Remove Constraints overview

Use the **Show / Remove Constraints** command to display the geometric constraints that are associated with sketch geometry. Use **Show / Remove Constraints** to:

- Remove specified constraints.
- List information about all geometric constraints.
- Interrogate and resolve over-constrained or conflicting conditions.
- Maintain design intent by checking for existing relationships to outside features or objects.

3

Where do I find it?

Toolbar	<p>(Modeling)</p> <p>Direct Sketch® Show / Remove Constraints </p> <p>(Sketch task environment)</p> <p>Sketch Tools® Show / Remove Constraints </p>
Menu	<p>(Modeling) Tools® Sketch Constraints® Show / Remove Constraints</p> <p>(Sketch task environment)</p> <p>Tools® Constraints® Show / Remove Constraints</p>



Show / Remove Constraints dialog box

Options	
List Constraints for	Controls which constraints are listed in the Show Constraints list window. Selected Object - Lets you display constraints for the object selected. Selected Objects - Lets you display constraints for multiple objects. All in Active Sketch - Shows all of the constraints in the active sketch. This is the default setting.
Constraint Type	Lets you filter constraints by type.
Include or Exclude	Determines whether the specified Constraint Type is the only type displayed in the list box (Include , which is the default) or the only type not displayed (Exclude).
Show Constraints	Lets you control the display of constraints in the list window. Options are: Explicit - Displays all constraints created explicitly or implicitly by the user, including all non-inferred coincident constraints, but excluding all inferred coincident constraints created by the system during curve creation. Inferred - Displays all inferred coincident constraints that are automatically created by the system during curve creation. Both - Displays both Explicit and Inferred types of constraints. Show Constraints list window - Lists the geometric constraints of the selected sketch geometry. The list is subject to the Explicit, Inferred or Both setting.
Remove Highlighted	Lets you remove one or more constraints by selecting them in the constraints list window and then choosing this option.
Remove Listed	Removes all of the listed constraints displayed in the Show Constraints list window.
Information	Displays information about all geometric constraints in the active sketch in the Information window. This option is useful if you want to save or print out the constraint information.

Dimensional Constraints overview

Use dimensional constraints, also called sketch dimensions, to establish the:

- Size of a sketch object.
- Relationship between two objects in a sketch.
- Relationship between two sketches.
- Relationship between a sketch and another feature.

Dimensional constraints display like drafting dimensions in that they have dimension text, extension lines, and arrows. However, dimensional constraints differ from drafting dimensions in that you can change the dimension value. This lets you control a feature derived from a sketch. Sketch dimensions also create an expression you can edit in the **Expressions** dialog box.

Where do I find it?

Toolbar	<p>(Modeling)</p> <p>Direct Sketch® Inferred Dimensions </p> <p>(Sketch task environment)</p> <p>Sketch Tools® Inferred Dimensions </p>
Menu	<p>(Modeling) Insert® Sketch Constraint® Dimension® Inferred Dimensions</p> <p>(Sketch task environment)</p> <p>Insert® Dimensions® Inferred Dimensions</p>

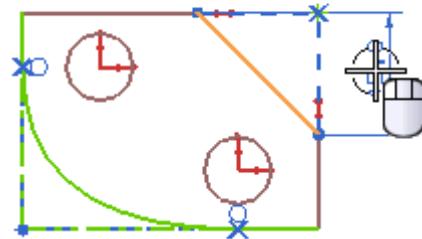
Sketch dimension types

Sketch Dimension types		
	Inferred	Infers a dimension type, based on the cursor position and the object(s) selected.
	Horizontal	Creates a distance constraint parallel to the XC axis between two points.
	Vertical	Creates a distance constraint parallel to the YC axis between two points.
	Parallel	Creates a distance constraint between two points.
	Perpendicular	Creates a perpendicular distance constraint from a line to a point.
	Diameter	Creates a diameter constraint for an arc or circle.
	Radius	Creates a radius constraint for an arc or circle.
	Angular	Dimensions an angle.
	Perimeter	Creates a dimensions for the collected length of selected curves.



Inferred Dimensions overview

Use the **Inferred Dimensions** command to create a dimensional constraint. NX infers the dimension type based on the objects you select and the location of the cursor.



3

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Inferred Dimensions (Sketch task environment) Sketch Tools® Inferred Dimensions
Menu	(Modeling) Insert® Sketch Constraint® Dimension® Inferred Dimensions (Sketch task environment) Insert® Dimensions® Inferred Dimensions

Create Inferred Dimensions

1. Optional: Set your annotation preferences.
2. On the **Sketch Tools** toolbar, click **Inferred Dimensions** .
3. In the graphics window, select the sketch object(s) you want to dimension.
4. Drag the dimension preview and click to place it on the sketch.
5. Click **Inferred Constraints** or the middle mouse button to exit the command.

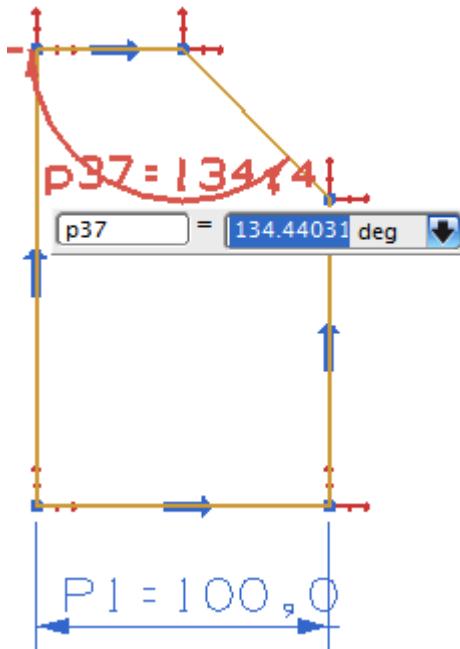


Inferred Dimensions allow for the creation of many different kinds of dimensions.

You can also create specific dimension types by selecting the dimension type from the **Inferred Dimensions** list.



In the following example, you can dynamically edit the expression name and value in the on-screen input box.



Edit sketch dimensions

Use this procedure to edit a sketch dimension using the on-screen input box.

- In the graphics window, double-click the dimension.



You can also right-click over a dimension and choose **Edit Value**.

- Edit the name or value in the on-screen input box.



To edit a formula value, click **Launch the formula editor** .

- Press Enter.



To use the **Dimensions** dialog box options, on the



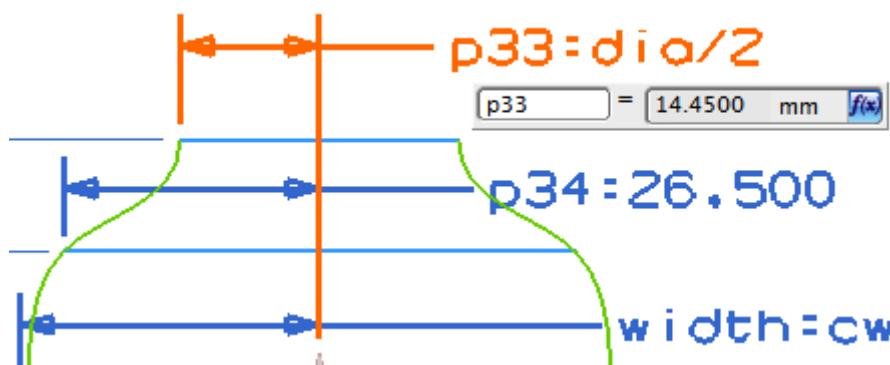
Dimensions dialog bar, click **Sketch Dimensions Dialog** .

To edit the dimension position, drag the dimension.



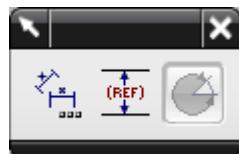
In the following example you can edit either the name of the sketch dimension, a constant value, or a formula.

The name and value of a dimension may also be edited by using the **Expressions** dialog box. As dimensions are edited, the constraints are evaluated and the geometry is modified.

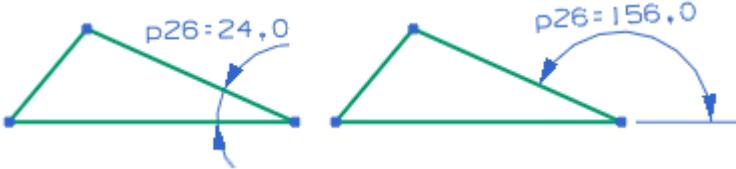


Dimensions dialog bar

When you select a dimension command from the **Dimensions Drop-down** , the **Dimensions** dialog bar appears.

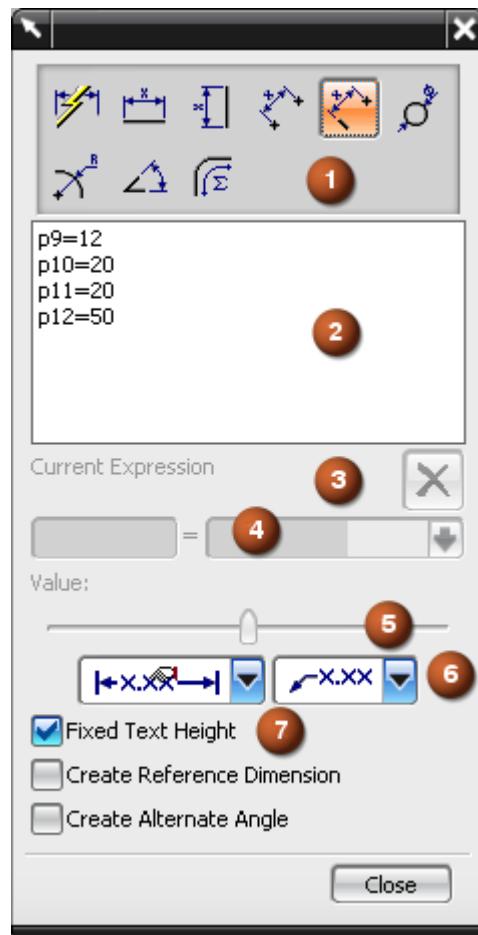


	Dimensions	Opens the Dimensions dialog box.
	Create Reference Direction	Activate this option to create reference dimensions. NX remembers the setting during the current session and applies it to subsequent dimensions you create.
	Create Alternate Angle	Activate this option when you want NX to calculate the maximum dimension between sketch curves. The figure below shows the same dimension with this option off (left) and on (right):



Dimensions dialog box

When **Dimensions** is selected from the **Dimensions** dialog bar, the **Dimensions** dialog box opens.



1. Lets you specify commands for creating inferred or explicit dimensions.
2. Lists the names and values of all dimensions in the current sketch.
3. Lets you delete the selected dimension.
4. Lets you edit the name and value of a selected dimension.
5. Lets you change the value of the selected dimensional constraint by dragging the slider.
6. Lets you specify how NX places dimensions:
7. Maintains the text of dimensions at a constant size when you zoom a sketch in or out.



Show Dimensions overview

Use the **Show Dimensions** command to display feature dimensions. To work with feature dimensions in the graphics window you can:

1. Right-click a feature in the **Part Navigator** or in the graphics window and choose **Show Dimensions**.
2. Double click a dimension to edit.
3. Use **Refresh** or press **F5** to hide dimensions.

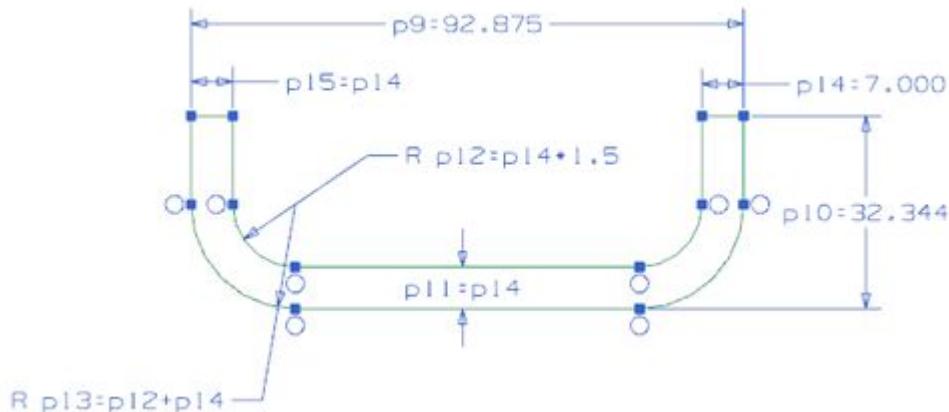


Any edits made to feature dimension will update immediately, including any dimensions or sketch data that has a relationship to the edited dimension.

Activities: Create constraints

In the *Sketching* section, do the activities:

- *Constrain a u-shaped profile*



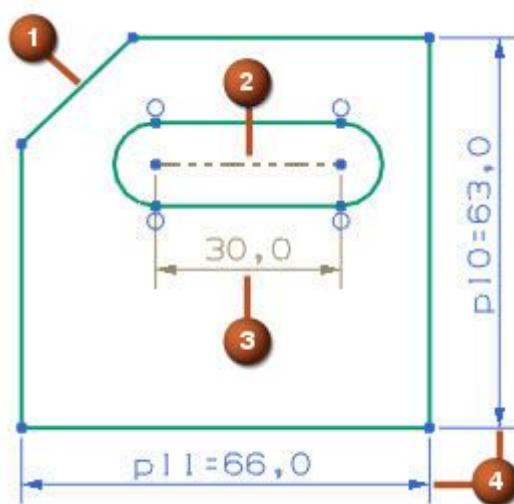


Convert To/From Reference overview

Use the **Convert To/From Reference** command to convert sketch curves from active to reference, or dimensions from driving to reference.

Downstream commands do not use reference curves and reference dimensions do not control sketch geometry.

By default NX displays reference curves in **Phantom** line font:



1. Active curves
2. Reference curve
3. Reference dimension
4. Driving dimensions

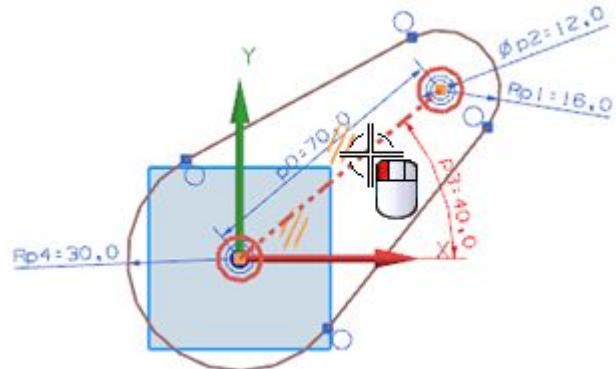
Where do I find it?

Toolbar	(Modeling) Direct Sketch® Convert To/From Reference  (Sketch task environment) Sketch Tools® Convert To/From Reference 
Menu	(Modeling) Tools® Sketch Constraints® Convert To/From Reference (Sketch task environment) Tools® Constraints® Convert To/From Reference
Shortcut Menu	Right-click a dimension ® Convert to Reference or Convert to Driving Right-click a curve ® Convert to Reference or Convert to Active

Activities: Sketch Constraints

In the *Sketching* section, do these activities:

- *Solve an over-constrained sketch condition*



Summary: Sketching

This lesson introduced the concept of creating a sketch.

In this lesson you:

- Created sketches directly in Modeling, on solid faces, and a Datum CSYS.
- Created and edited curves in a sketch.
- Created inferred and explicit geometric constraints.
- Created and edited dimensional constraints.

Lesson

4 *Constraining and using sketches*

Purpose

This lesson describes creating and modifying sketch constraints and additional sketch editing methods.

Objectives

4

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

- Auto create and display constraints.
- Constrain the perimeter of a sketch.
- Animate your sketch for movement visualization.
- Determine an alternate solution.
- Attach and Reattach a sketch.
- Mirror sketch curves and make curves symmetric.
- Create a linear pattern of curves.

Edit sketches with drag

You can use drag to edit the location and shape of your sketch. You can drag unconstrained sketch curves or points in unconstrained directions.

You can also use the **Dimensions** dialog box to edit a dimension by using the value slider.

Techniques for editing sketch curves with drag.

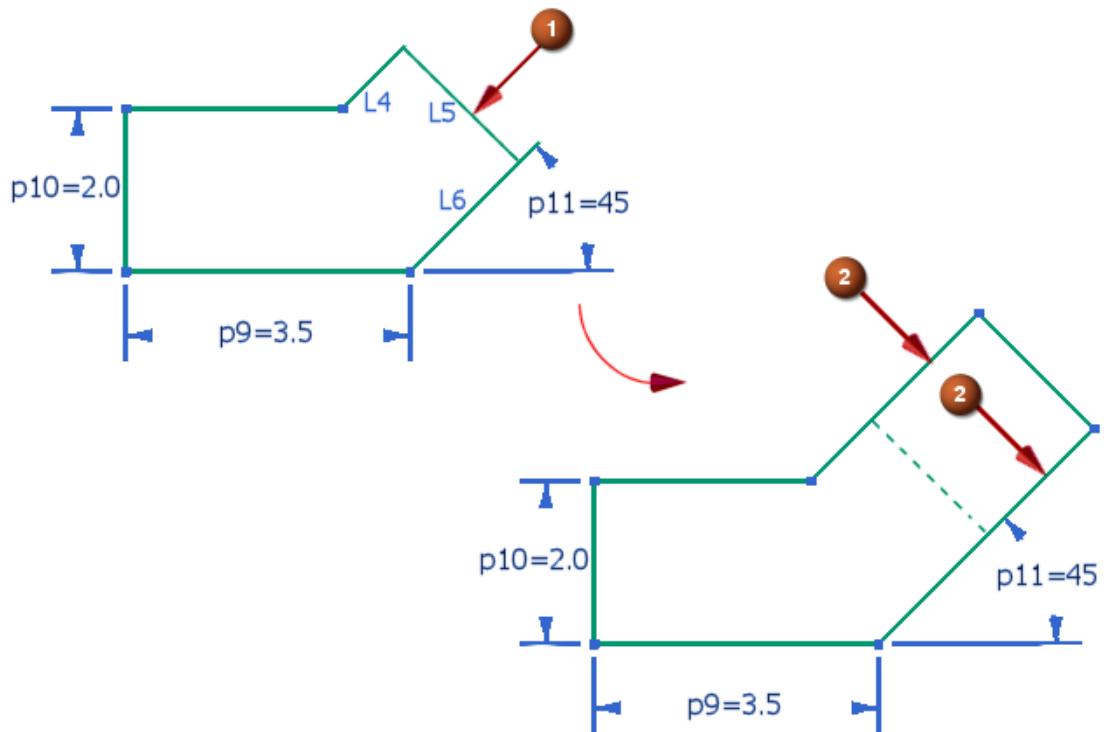
- There is no dialog box for this command. You can just drag selected objects in the graphics window.
- You can drag sketch curves that are partially constrained or unconstrained.
- You can drag fully constrained sketches if they have not yet been positioned.

When you drag constrained curves they are scaled as necessary to preserve the constraints.

- You can apply Inferred constraints if you drag the free end of a line.



In the following example, L5 (1) is being dragged while L4 and L6 (2) stretch. L5 is constrained so it maintains its angular and length relationship.



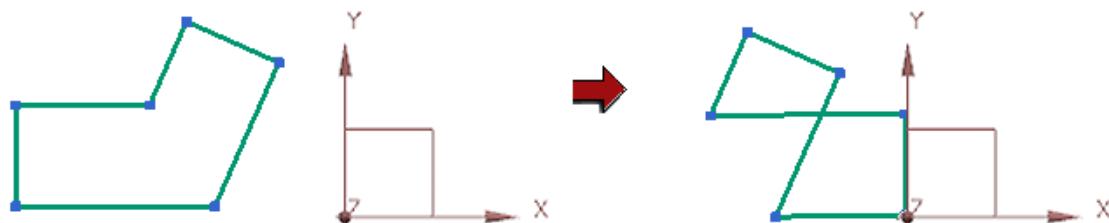
Drag to assist sketch constraints

You can drag curves to approximate the correct location before you constrain them.

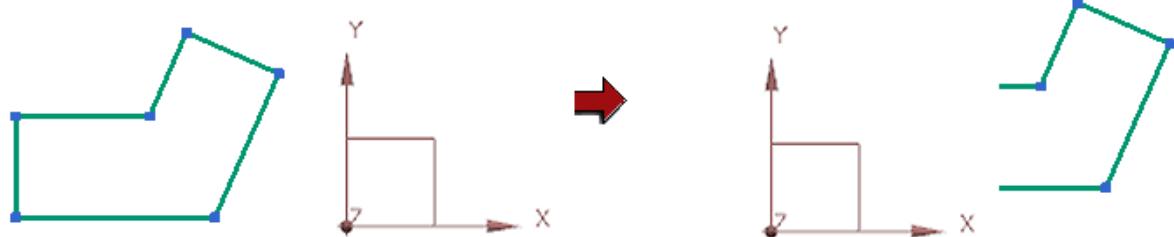
This is useful when constraining curves at their original location distorts the sketch, making it difficult to continue.



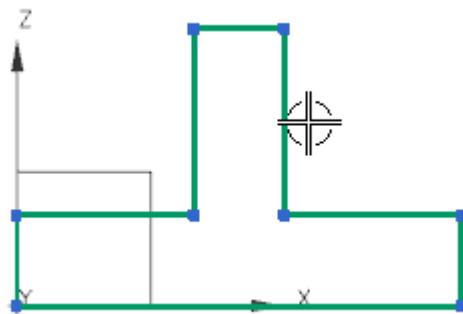
The following example shows the distortion that can be caused when you attempt to drag objects with too many applied constraints.



The following example shows the result of dragging an entire unconstrained profile from one quadrant to another.



Copy, move, and edit sketch objects



To	Do This
Move curves, points, or dimensions	Drag the curves, points, or dimensions
Move curves or points vertically or horizontally	Press Shift and drag the curves or points
Copy curves or points	Press Ctrl and drag the curves or points
Copy curves or points vertically or horizontally	Press Ctrl+Shift and drag the curves or points
Edit objects	Double-click the objects
Choose a command	Right-click the objects



If you are in another command that requires a selection, you must exit the command before you perform these actions. To exit a command, press Esc.



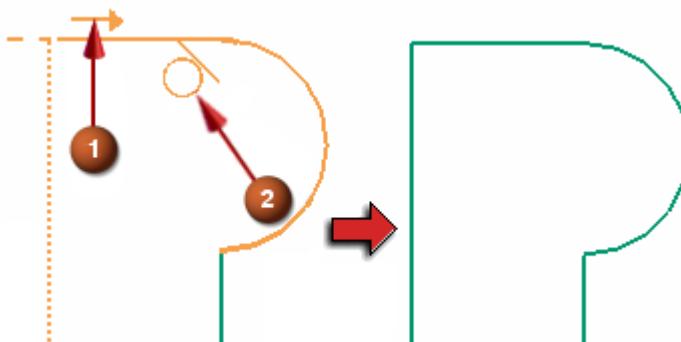
Create Inferred Constraints

Use the **Create Inferred Constraints** command to enable inferred constraints during curve construction.

By default this option is turned on and creates the constraints defined in the **Inferred Constraints** dialog box.

If you disable **Inferred Constraints**, you can take advantage of the constraints as you work, but the actual constraints are not stored in your file.

In the following example, the **Create Inferred Constraints** button is turned off. The tangent and horizontal constraints are available during curve creation (1 and 2). But the constraints are deleted when the profile is completed.



4

When you preview a constraint, click the middle mouse button to lock the constraint and prevent the sketch curve from moving in any other direction.



To temporarily disable , hold the **Create Inferred Constraints** Alt key.

The settings for each sketch is saved in the part file.

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Create Inferred Constraints (Sketch task environment) Sketch Tools® Create Inferred Constraints
Menu	(Modeling) Tools® Sketch Constraints® Create Inferred Constraints (Sketch task environment) Tools® Constraints® Create Inferred Constraints



Auto Dimension

Use the **Auto Dimension** command to create dimensions on selected curves and points according to a set of rules.

You can apply the following rules in any order.

- **Create Horizontal and Vertical Dimensions on Lines**
- **Create Dimensions to Reference Axes**
- **Create Symmetric Dimensions**
- **Create Length Dimensions**
- **Create Adjacent Angles**

Driving and Automatic Dimensions

You can create two types of dimensions.

Driving

Creates a dimension based on an expression. You can convert driving dimensions to reference dimensions.

Automatic

This type of dimensional constraint can only be created by auto dimensioning. Automatic dimensions remove degrees of freedom from the sketch, and act like a constant length or angle constraint with a value.

- When you drag sketch curves, automatic dimensions are relaxed, allowing you to shape the sketch. After the drag, the new shape will be constrained by the automatic dimensions again.
- If you add a constraint that conflicts with an automatic dimension, the automatic dimension is deleted.
- You can convert automatic dimensions into driving or reference dimensions.

All dimensions are displayed in the **Part Navigator** when **Timestamp Order** is turned off in the **Unused Items® Sketch® Curves and Dimensions® Dimensions** folder.

Why should I use it?

In Modeling, use this command to assist in creating a fully constrained sketch by removing all degrees of freedom from the selected curves.

In Drafting, use this command to fully dimension selected sketch curves in a drawing.

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Auto Dimension  (Drafting and Sketch task environment) Sketch Tools® Auto Dimension 
Menu	(Modeling) Tools® Sketch Constraints® Auto Dimension (Drafting and Sketch task environment) Tools® Constraints® Auto Dimension



Auto Constrain

The **Auto Constrain** command creates specific multiple geometric constraint types to selected sketch objects. The selected geometry is analyzed based on the command settings and the constraints are applied to your sketch.

This can be especially useful when you add geometry to the active sketch, particularly if that geometry was imported from a different CAD system. You can also use this to apply a single constraint type to multiple sketch objects, for example apply a tangent constraint to multiple fillets and their adjacent curves.

Where do I find it?

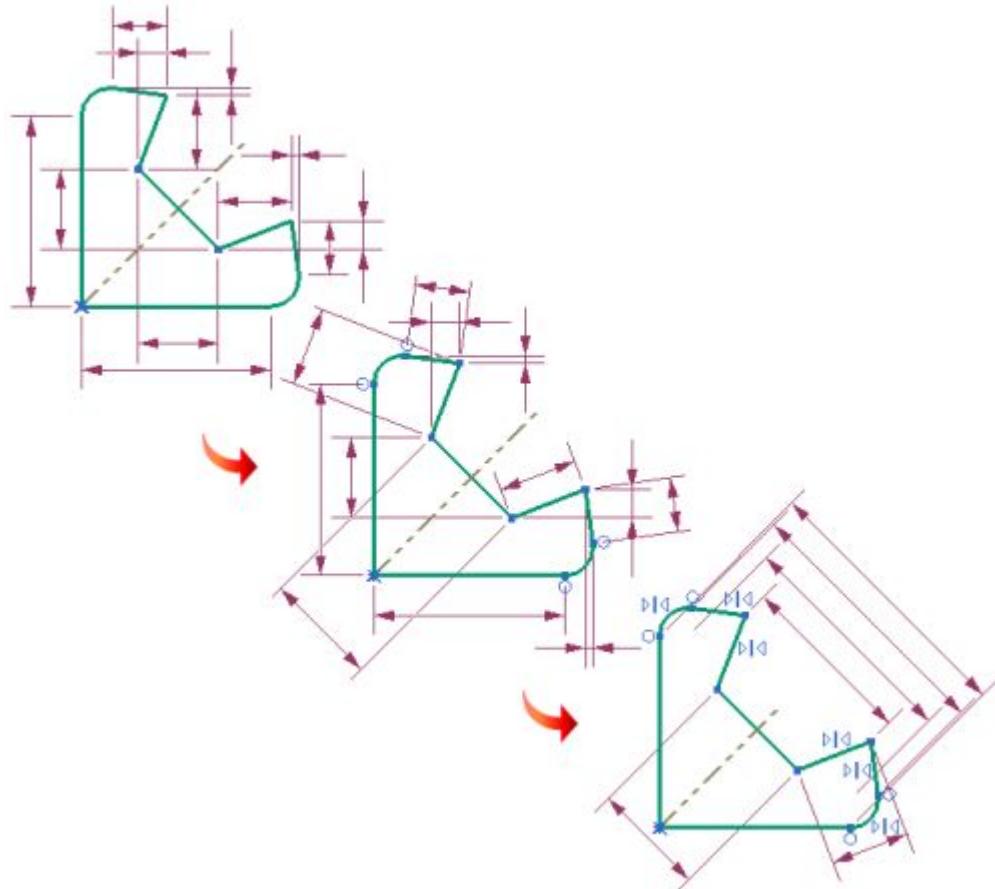
4

Toolbar	(Modeling) Direct Sketch® Auto Constrain (Sketch task environment) Sketch Tools® Auto Constrain
Menu	(Modeling and Sketch task environment) Tools® Constraints® Auto Constrain

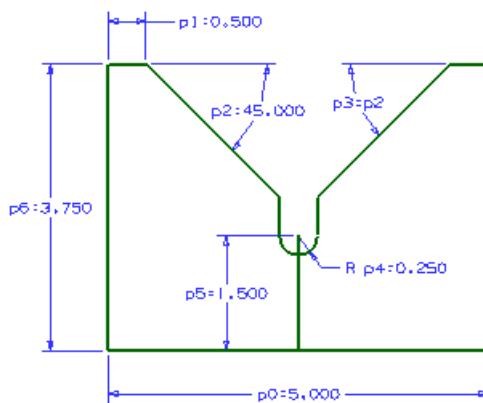
Activities: Auto and Perimeter constraints

In the *Using and constraining sketches* section, do the activity:

- *Auto Dimensioning Rules*



- *Create auto constraints*





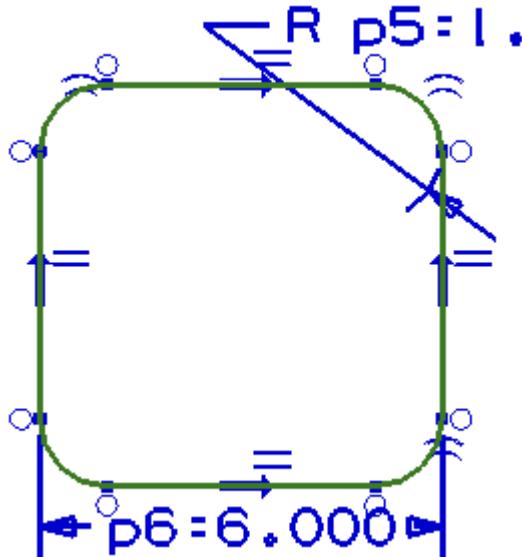
Sketch Animate Dimension

The **Animate Dimension** command dynamically displays the effects of varying a given dimension over a specified range. Any geometry affected by the selected dimension is also animated.



Unlike Drag, Animate Dimension does not change the sketch dimension value. When the animation is finished, the sketch returns to its original state.

4

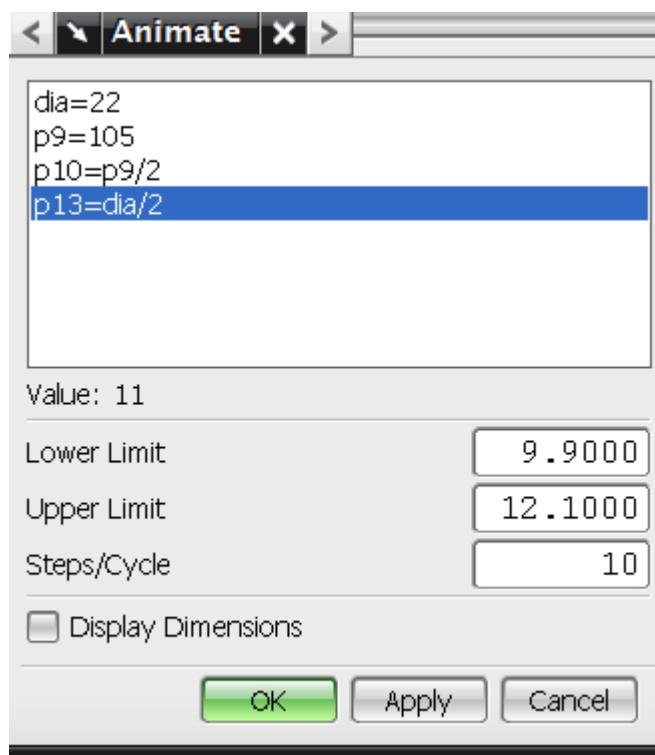


Where do I find it?

Toolbar Direct Sketch® Animate Dimension  (Sketch task environment)	(Modeling) Sketch Tools® Animate Dimension 
Menu Tools® Sketch Constraints® Animate Dimension (Sketch task environment) Tools® Constraints® Animate Dimension	

Animate Dimension options

- **Value** is the value of the currently selected dimension.
- **Lower Limit** defines the smallest value the dimension is during the animation.
- **Upper Limit** defines the largest value the dimension is during the animation.
- **Steps/Cycle** the number of times the dimension value changes (equal size increments) when it moves from the upper limit to the lower limit (or vice versa).
- **Display Dimensions** displays the original sketch dimensions during the animation.

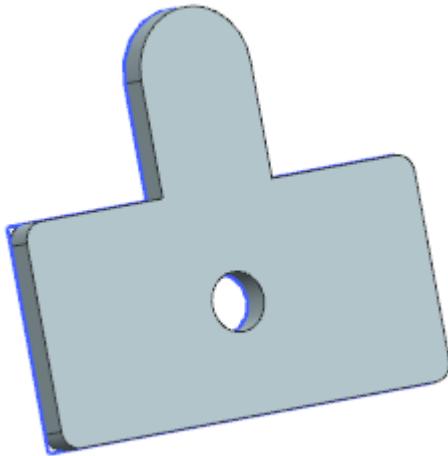


Activity: Animate dimension

In the *Constraining and using sketches* section, do the activity:

Experiment with sketch dimensions

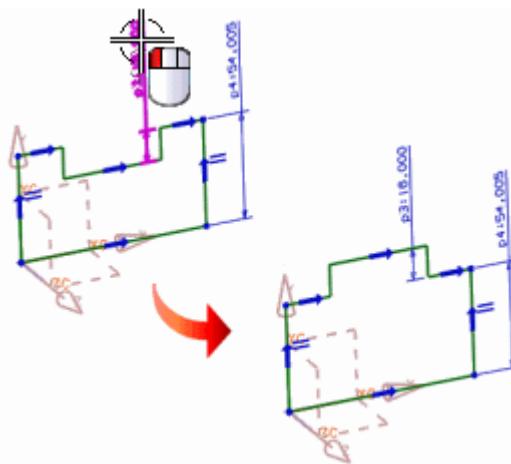
4





Alternate Solution overview

Use the **Alternate Solution** command to display alternate solutions for both dimensional and geometric constraints, and select a result. The example below shows how the geometry changes when you choose Alternate Solution and select a dimension.



Where do I find it?

Toolbar	(Modeling) Direct Sketch® Alternate Solution  (Sketch task environment) Sketch Tools® Alternate Solution 
Menu	(Modeling) Tools® Sketch Constraints® Alternate Solution (Sketch task environment) Tools® Constraints® Alternate Solution

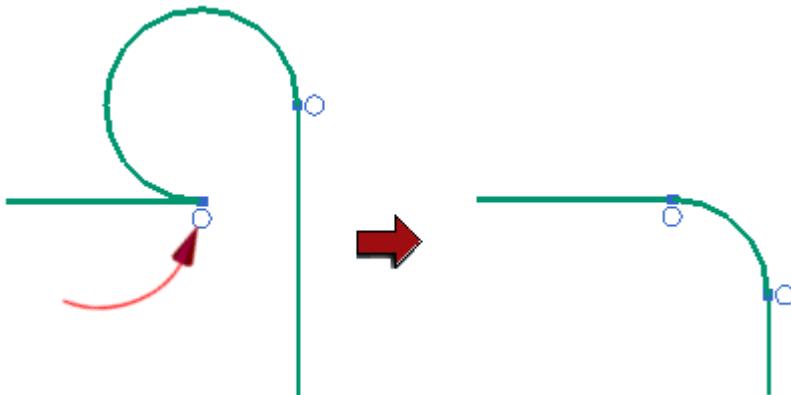
4

Use Alternate Solution on tangent constraints

1. On the **Sketch Tools** toolbar, click **Alternate Solution** .
2. For **Object 1, Select Linear Dimension or Geometry**, select either the line or arc.
 -  If either curve does not have a tangency constraint you will not be able to select the object. But you can select the geometry in any order.
3. For **Object 2, Select Tangent Geometry**, select the corresponding line or arc.
 -  If the range of movement is limited to a single direction this step may not be necessary.
4. Click **Close**.



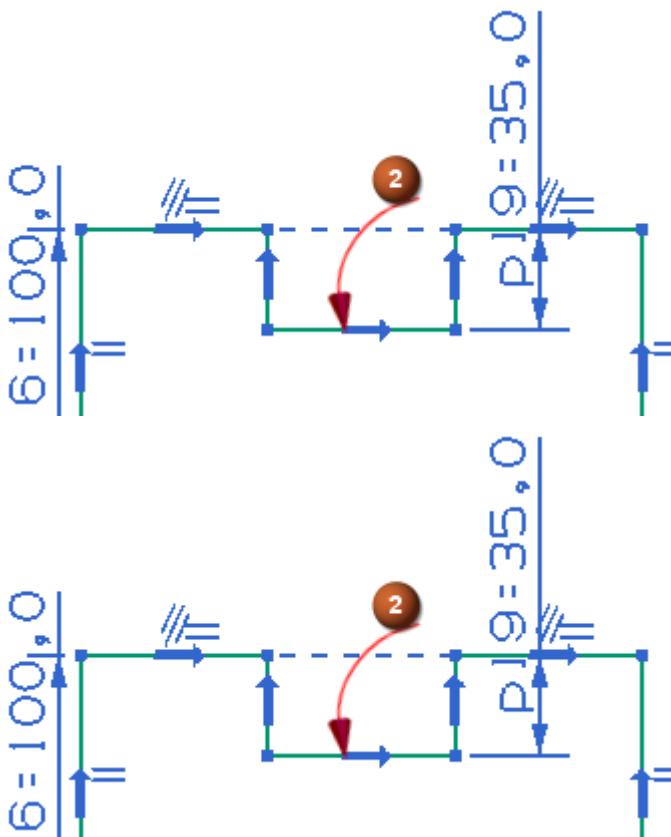
In the following example, Alternate Solution was used to properly create the fillet by modifying the tangent constraint.



Use Alternate Solution on a dimension

1. On the **Sketch Tools** toolbar, click **Alternate Solution** .
2. For **Object 1, Select Linear Dimension or Geometry**, select a sketch dimension.
💡 The change is immediate. Simply select the dimension again to restore the previous solution.

 In the following example, the sketch dimension **p19** (1) was selected and an alternate solution found. In this case the alternate solution was to reverse the direction of the sketch curves (2).

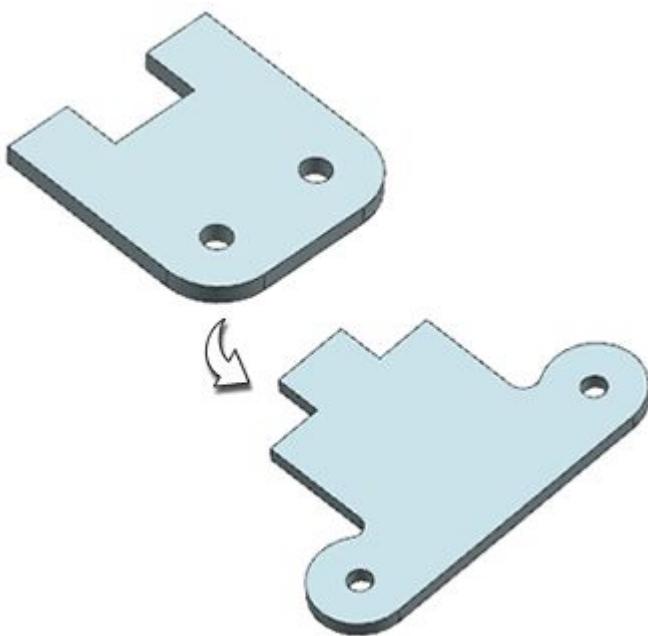


Activities: Alternate solutions

In the *Constraining and using sketches* section, do the activity:

Experiment with alternate solutions

4

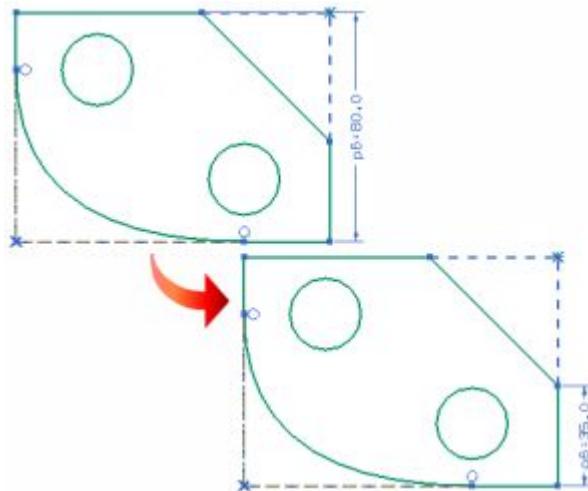




Attach Dimension overview

Use the **Attach Dimension** command to attach an existing dimension to different objects. When you attach a dimension you can:

- Retain the expression value and resize the target geometry to match it.
- Change the expression value to match the target geometry.



4

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Attach Dimension  (Sketch task environment) Sketch Tools® Attach Dimension 
Menu	(Modeling) Tools® Sketch Constraints® Attach Dimension (Sketch task environment) Tools® Constraints® Attach Dimension
Shortcut Menu	Right-click a dimension and choose Attach Dimension

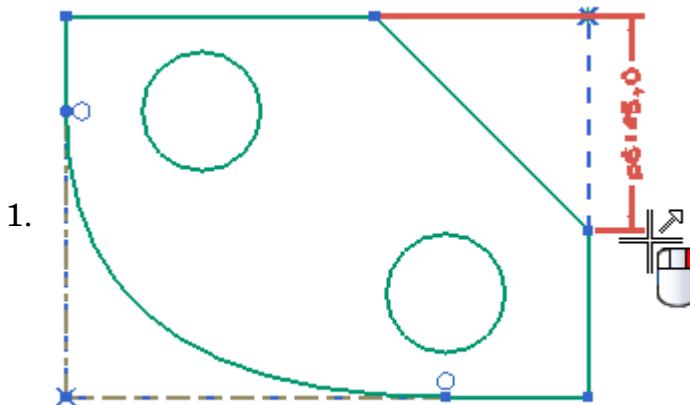


Attach a Dimension to different geometry

This example shows you how to attach a dimension to a different curve.

Note that you can optionally measure the target geometry and assign its geometric value to the dimension variable.

4



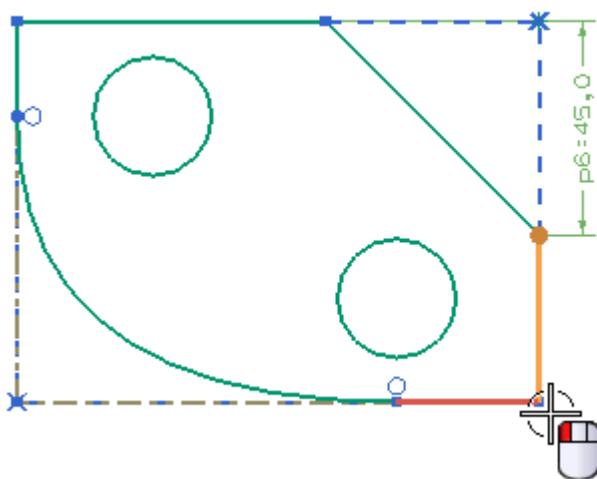
Right-click the dimension leader line you want to attach and choose **Attach Dimension**.

2.

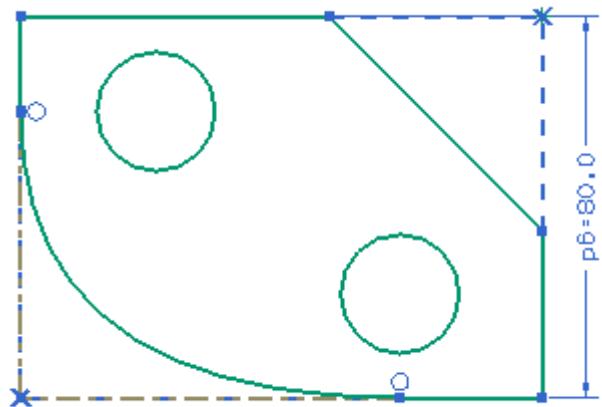


In the **Settings** group, set the **Expression Mode** to **Remove Expression**, **Measure Geometry**.

3.



Left-click the new curve.





Reattach Sketch overview

Use the **Reattach** command to:

- Move a sketch to a different plane, face, or path.
- Switch a **Sketch in Place** to a **Sketch on Path** and vice versa.
- Change the location of a sketch on path along the path to which it is attached.
- Specify a new horizontal or vertical reference.

Where do I find it?

4

Toolbar	(Modeling) Direct Sketch® Reattach  (Sketch task environment) Sketch Tools® Reattach
Menu	(Modeling and Sketch task environment) Tools® Reattach

Reattach a sketch on plane

Use this procedure to reattach a sketch to a planar surface.

1. Open the target sketch for editing.

2. On the **Direct Sketch** toolbar, click **Reattach** .

3. Select the new target planar surface.



The preview sketch CSYS moves from the original face to the new face.

4. Optional: Expand the **Sketch Orientation** group, from the **Reference** list, choose either **Horizontal** or **Vertical**.

5. Optional: Click **Select Reference** and define the new reference direction

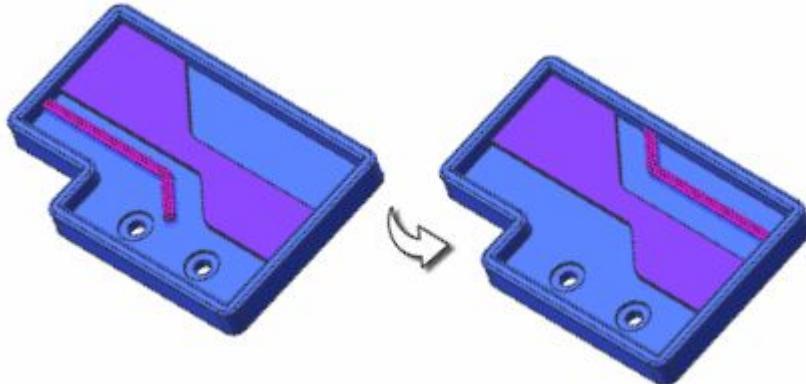


6. Click **OK**.

Activities: Reattach sketches

In the *Constraining and using sketches* section, do the activity:

- *Reattach a sketch to a new face*

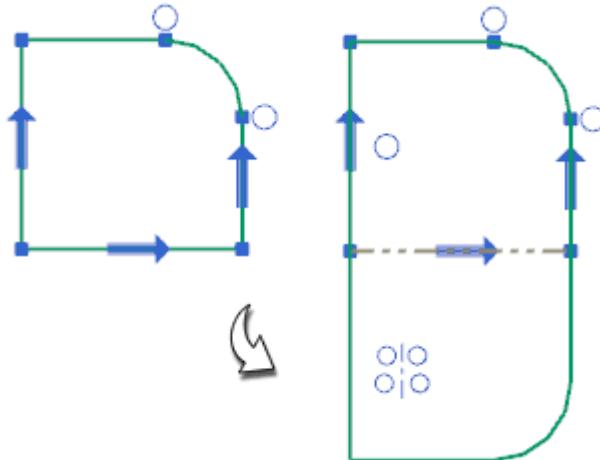


4



Mirror Curve overview

Use the **Mirror Curve** command to make a mirrored copy of sketch geometry through a specified sketch line. NX applies mirror geometric constraints to all the geometry.



4

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Mirror Curve (Drafting and Sketch task environment) Sketch Tools® Mirror Curve
Menu	(Modeling and Drafting) Insert® Sketch Curve® Mirror Curve (Sketch task environment) Insert® Curve from Curves® Mirror Curve

Mirror sketch curves

1. On the **Direct Sketch** toolbar, click **Mirror Curve** .
2. Select the curves you need to mirror.
3. For the **Centerline**, select an existing sketch curve.
4. Click **OK**.



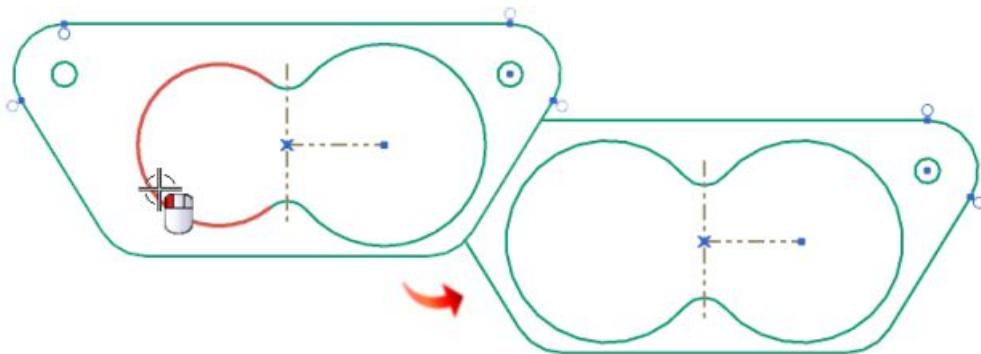
Make Symmetric

Use the **Make Symmetric** command to constrain two points or curves to be symmetric about a centerline in a sketch.

You can apply the symmetry constraint between two objects of the same type:

- Lines
- Arcs
- Circles

You can also make different point types symmetric. For example, you make the end of a line and the center of an arc symmetric about a line.



Why should I use it?

Use this command when you create or edit a sketch and you want to control the position of existing sketch geometry to be symmetric to a centerline.

Where do I find it?

	(Modeling) Direct Sketch® Make Symmetric
Toolbar	(Drafting and Sketch task environment) Sketch Tools® Make Symmetric
	(Modeling and Drafting) Insert® Sketch Constraint® Make Symmetric
Menu	(Sketch task environment) Insert® Make Symmetric



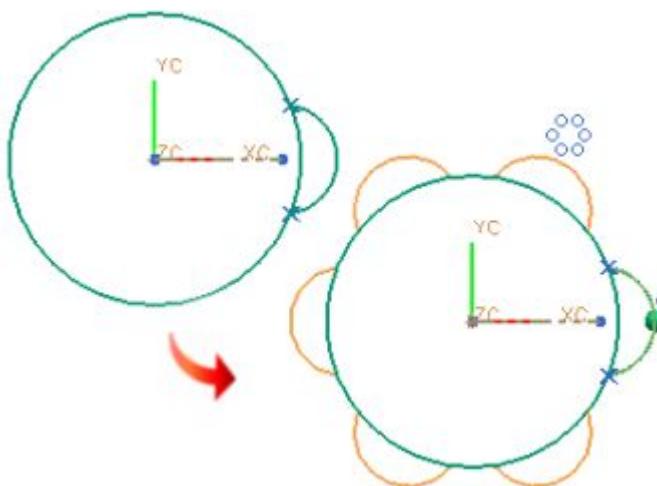
Pattern Curve

Use the **Pattern Curve** command to pattern edges, curves, and points that are parallel to the sketch plane.

The available types are:

- **Linear Pattern**
- **Circular Pattern**

This command also creates a pattern constraint that can be modified when you double-click one of the patterned curves.



4

Circular Pattern with 5 Instances

Where do I find it?

Toolbar	(Modeling) Direct Sketch® Pattern Curve (Drafting and Sketch task environment) Sketch Tools® Pattern Curve
Menu	(Modeling and Drafting) Insert® Sketch Curve® Pattern Curve (Sketch task environment) Insert® Curve® Pattern Curve

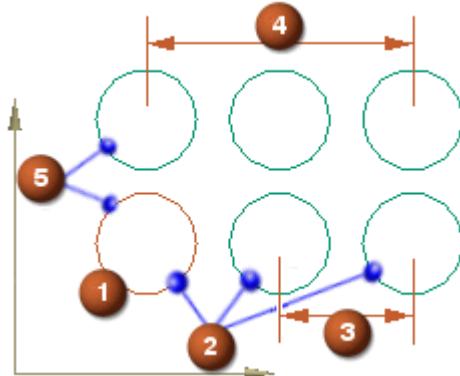


Pattern Curve linear associative options

This topic describes **Linear Pattern Curve** options that are available if you enable **Create Inferred Constraints**.

1. Selected curve for pattern.
2. **Direction 1** sets the number of objects in the pattern for direction one, in this case the **Count** is set to **3**.
3. **Pitch Distance** sets the distance between each copy of the selected curves.
4. **Span Distance** sets the distance from the selected curve to the last curve in the pattern.
5. **Direction 2** sets the number of objects in the pattern for direction two.
6. **Create Pitch Expressions** creates expressions for the pattern count and pitch. The expressions are named `Pattern_p#`.

4

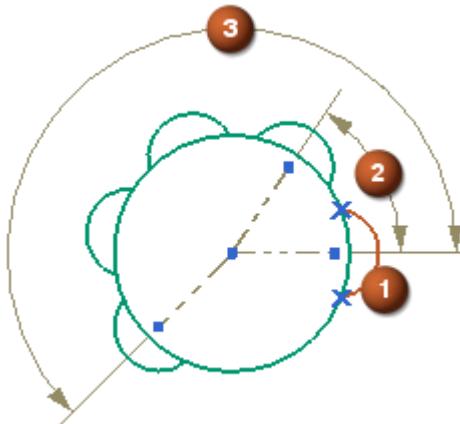




Pattern Curve circular associative options

This topic describes **Circular Pattern Curve** options that are available if you enable **Create Inferred Constraints**.

1. Selected curve for pattern sets the angle from the selected curve to the last curve in the pattern.
2. **Pitch Angle** sets the angle between each copy of the selected curves.
3. **Span Angle** sets the number of objects in the pattern, in this case the **Count** is set to **5**.
4. **Create Pitch Expressions** creates expressions for the pattern count and pitch. The expressions are named `Pattern_p#`.



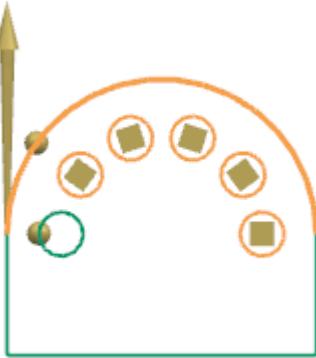


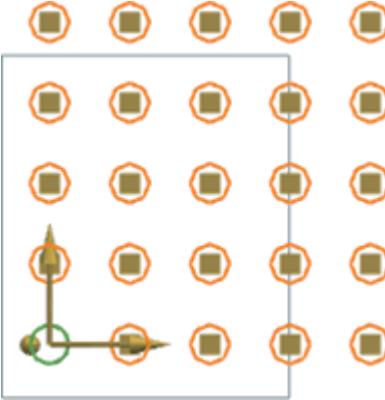
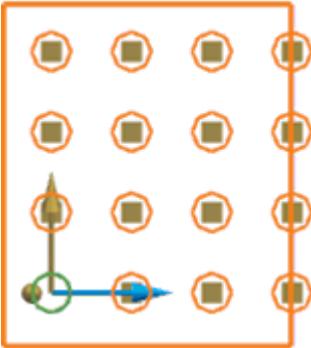
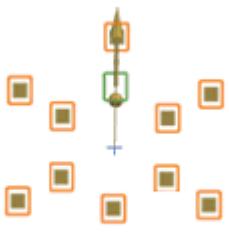
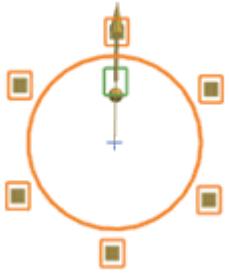
Pattern Curve non-associative options

This topic describes additional **Pattern Curve** options only available if you disable **Create Inferred Constraints**. These options are especially useful when you add sketch curves to a drawing.

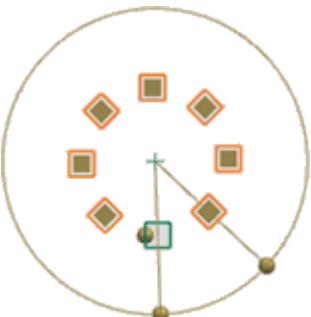
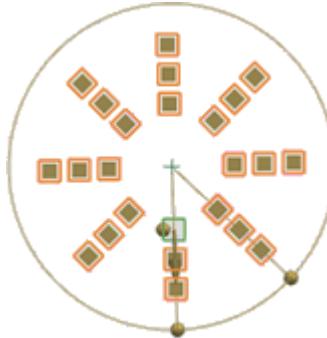
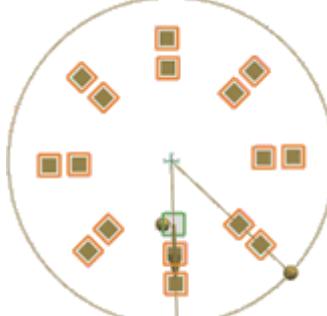
Pattern Definition

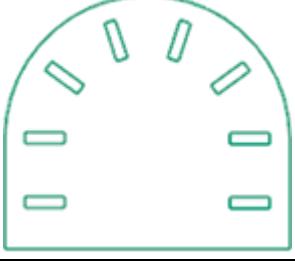
Layout Along – A pattern whose direction is defined by a continuous chain of curves or edges, with instances distributed along the chain according to the spacing parameters.

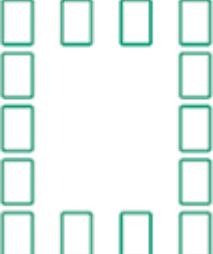
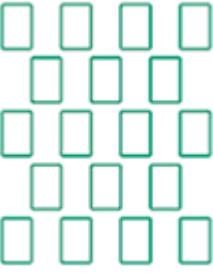
	<ul style="list-style-type: none">• Count = 6• Location = % Arc Length• % Pitch Distance = 20• Orientation = Normal to Path
---	--

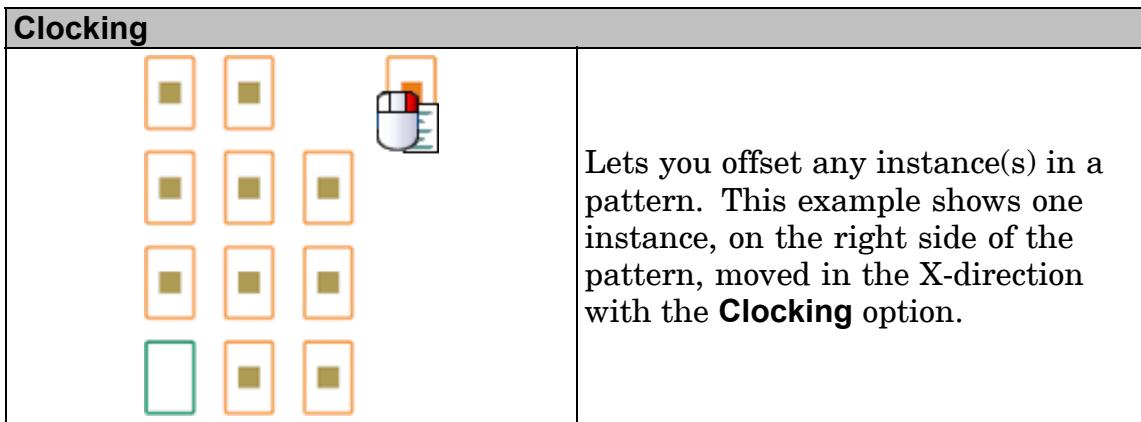
Boundary Definition	
Available for Linear and Circular layouts.	Lets you specify a curve or face as a Boundary that will clip pattern instances whose control points reach the boundary, or if they are inside the boundary.
	<ul style="list-style-type: none"> Layout = Linear Count = 5 and Pitch = 12 in two directions Boundary = None
	<ul style="list-style-type: none"> Boundary = Face Margin Distance = 4 All four face edges selected.
	<ul style="list-style-type: none"> Boundary Curve = None
	<ul style="list-style-type: none"> Boundary Curve = Internal Only Circle selected as the boundary. NX clips pattern instances inside the boundary.

4

Radial Direction	Available for Circular layouts.
	Lets you add concentric members to a pattern, and set the spacing.
	<ul style="list-style-type: none"> • Pitch Angle = 45 • Span Angle = 360 • Create Concentric Members = off
	<ul style="list-style-type: none"> • Create Concentric Members = on • Include First Circle = on • Count = 3 • Pitch = 3
	<ul style="list-style-type: none"> • Include First Circle = off

Orientation	
Available for all layouts.	Controls whether instances maintain a constant orientation, or follow an orientation based on either the pattern definition or the path.
	<ul style="list-style-type: none"> • Layout = Circular • Orientation = Fixed
	<ul style="list-style-type: none"> • Layout = Circular • Orientation = Follow Pattern
	<ul style="list-style-type: none"> • Layout = Along • Orientation = Normal to Path

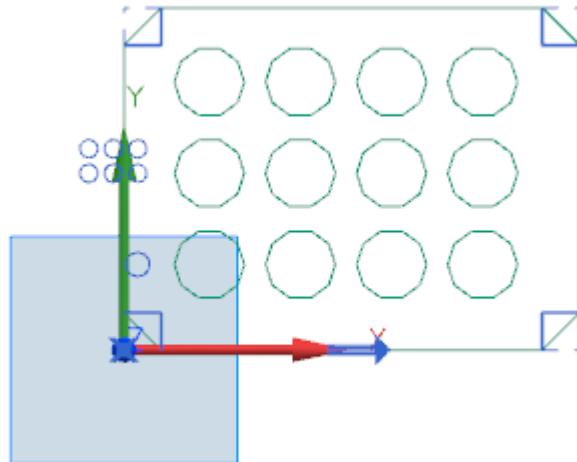
Pattern Settings	
	<p>Frame Only</p> <p>Available for Linear layouts. When Frame Only is enabled, NX creates only the outermost instances of a pattern.</p>
	<p>Stagger</p> <p>Available for Linear and Circular layouts. Lets you vary the locations of adjacent rows or columns for a Linear layout, and an angle for a Circular layout. The distance is a preset 1/2 pitch. This example shows a row stagger.</p>



Activities: Pattern curves

In the *Constraining and using sketches* section, do the activities:

- *Create a linear pattern in two directions*



Sketch evaluation and update techniques

You would use this process to control when the system evaluates a sketch for updating.

- Click **Delay Evaluation** .
 - Create or edit sketch constraints or curves.
 - Click **Evaluate Sketch** .
 - Optional: To update the model while still in the sketch, on the **Sketch Tools** toolbar, click **Update Model** .
-  **Update Model** is not available when you use **Edit with Rollback** to open your sketch for editing. However, all models are updated as soon as you exit the sketch.

Summary: Using and constraining sketches

Sketching can be used to plan, edit, and manipulate your design through every phase of the design cycle.

In this lesson you:

- Auto created and displayed constraints.
- Constrained the perimeter of a sketch.
- Animated your sketch for movement visualization.
- Applied alternate solutions to obtain appropriate profiles.
- Reattached sketches.
- Mirrored curves and made curves symmetric.
- Created a pattern of curves.

4

Lesson

5 *Datum features*

Purpose

This lesson introduces the datum plane, datum axis, and datum CSYS reference features.

Objectives

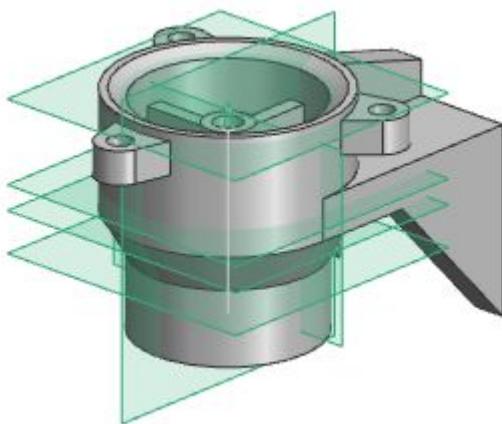
Upon completion of this lesson, you will be able to:

- Create a datum plane.
- Create a datum axis.
- Use datum features to position other features.
- Create a datum CSYS.



Datum Plane overview

Use the **Datum Plane** command to create a planar reference feature to help define other features, such as swept bodies and features at angles to the faces of target solids.



5

Datum planes can be relative or fixed.

Relative datum planes

Relative datum planes reference curves, faces, edges, points, and other datums. You can create relative datum planes across multiple bodies.

Fixed datum planes

Fixed datum planes do not reference other geometry. Use any of the relative datum plane methods to create fixed datum planes by clearing the **Associative** box in the **Datum Plane** dialog box.

Where do I find it?

Application	Modeling
Toolbar	Feature® Datum/Point Drop-down® Datum Plane
Menu	Insert® Datum/Point® Datum Plane
Shortcut menu	Right-click a planar face® Datum Plane

Datum plane types

Select a plane type from the **Type** option list.



You can right-click the sizing handles, direction arrows, and points to choose many of the following options.

When you edit a datum plane, you can change its type, defining objects, and associative status.

Inferred Determine the best plane type to use based on objects you select.

At Angle Create a datum plane using a specified angle.

At Distance Create a datum plane parallel to a planar face or another datum plane at a distance you specify.

Bisector Create a datum mid way between two selected planar faces or datum planes using the bisected angle.

Tangent Create a datum plane tangent to a non-planar surface, and optionally a second selected object.

Datum plane options



Alternate Solution

Cycle through the possible different solutions for the plane, when an alternate solution to the previewed datum plane is available.



Reverse Plane Normal

Reverse the direction of the plane normal.

Associative

Clear this check box to create a fixed datum plane. If you later edit a non-associative datum plane, it appears in the **Type** list as **Fixed**.

Applications for datum planes

- To define a sketch plane.
- To serve as the planar placement face for the creation of features with predefined shapes.
- As a target edge for positioning features such as holes.
- For the mirror plane when using the **Mirror Body** and **Mirror Feature** commands.
- To define the start or end limits when creating extruded and revolved features.
- To trim a body.
- To define positioning constraints in assemblies.
- To help define a relative datum axis.



Create a datum plane using offset

This example shows how to create a datum plane that is offset from an inferred plane using two lines.

Step 1



Click **Datum Plane**



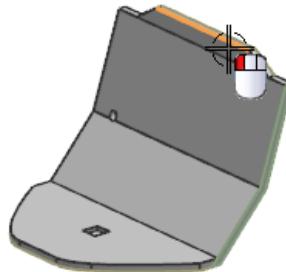
Step 2



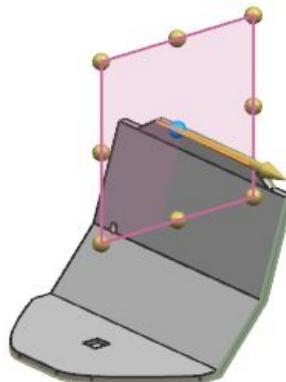
From the **Type** list, select
Inferred



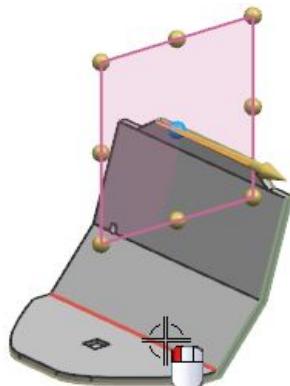
Step 3



Select the first linear edge to define the plane.



Step 4



Select the second linear edge to define the plane.

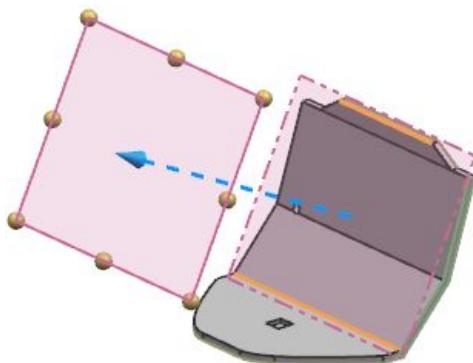
Step 5



In the **Offset** group, select the **Offset** check box.

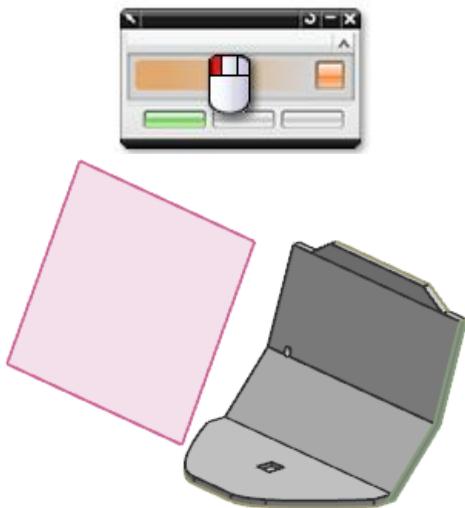
Verify the direction of the desired offset.

Step 6



Drag the direction arrow the desired value of the offset, or enter it in the offset **Distance** box.

Step 7



Click **OK**.

If autocomplete is available,
you can choose another
command without clicking
OK.



Create a datum plane midway between planar faces

The following example shows how to create a datum plane that bisects the angle formed by two planar faces.

Step 1



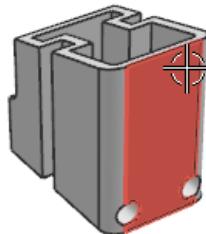
Click **Datum Plane**

Step 2



From the Type list, select
Bisector

Step 3



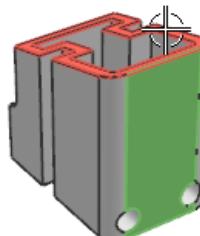
Select the front planar face.

5

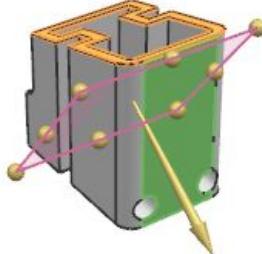
Step 4



Click the middle mouse button
to advance to the next step.

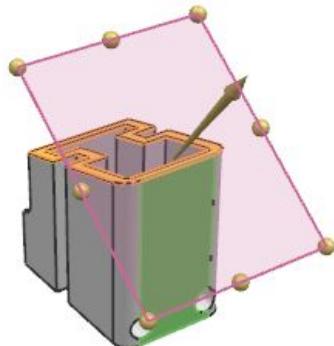


Step 5



Select the top planar face.

Step 6



(Optional) In the **Plane Orientation** group, click **Alternate Solution** to get a different solution to the selections made.

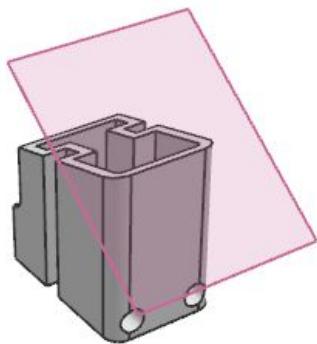
In this example, **Alternate Solution** was clicked once.

Step 7



Click **OK**.

If autocomplete is available, you can choose another command without clicking **OK**.





Create a datum plane at an angle

The following example shows how to create a datum plane at an angle of 115 degrees from a reference plane.

Step 1



Click **Datum Plane**



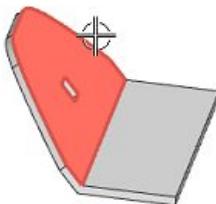
Step 2



From the Type list, select **At Angle**



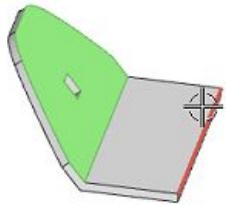
Step 3



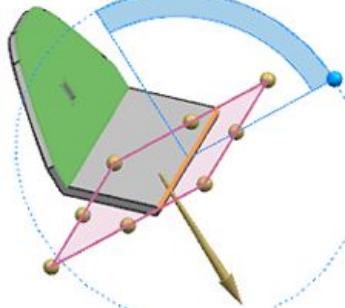
Select the front angled face.

5

Step 4



Select the top right linear edge of the part.

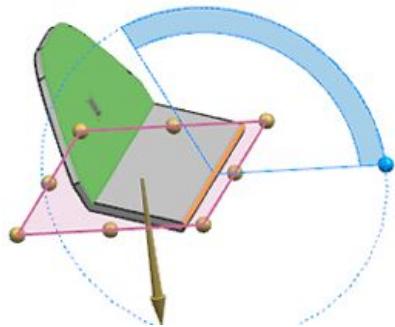


Step 5



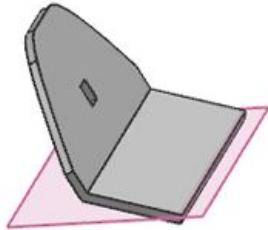
From the **Angle Option** list, select **Value**.

Step 6



Drag the angle handle until the value shows **115** degrees.

Step 7



Click **OK**.

If auto-complete is available, you can choose another command without clicking **OK**.



Create a datum plane through three points

The following example shows how to create a datum plane that contains three specified points.

Step 1



Click **Datum Plane**



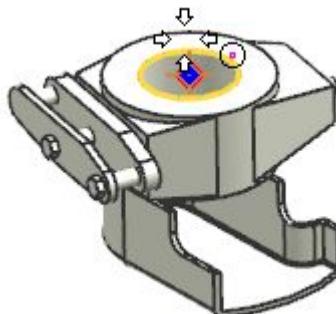
From the Type list, select **Curves and Points**.

Step 2



In the **Curves and Points** Subtype group, from the Subtype list, select **Three Points**.

Step 3

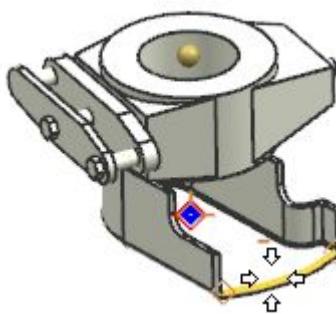


This example will use two arc centers and one end point.

Specify the first point.

Select the arc center in the top face.

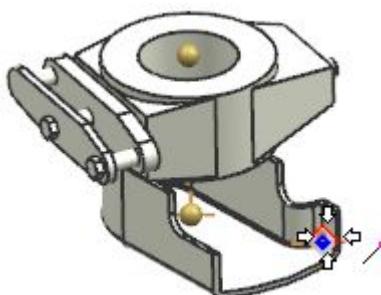
Step 4



Specify the second point.

Select the arc center on the bottom inside face.

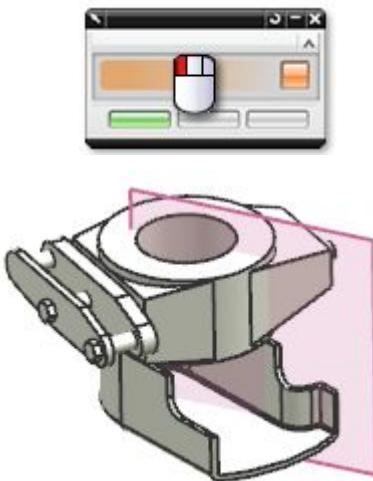
Step 5



Specify the third point.

Select an End Point on the bottom surface.

Step 6

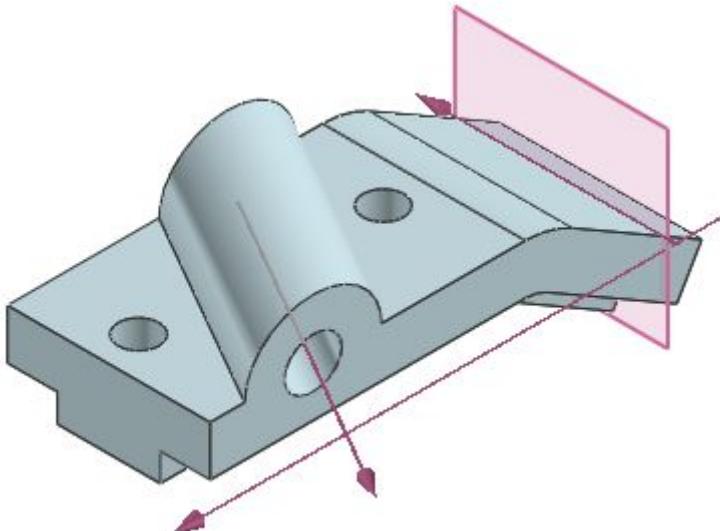


Click **OK**.

If autocomplete is available,
you can choose another
command without clicking
OK.

Datum Axis overview

Use the **Datum Axis** command to define linear **reference objects** to help you create other objects, such as datum planes, revolved features, extruded features, and circular **arrays**.



5

Datum axes can be relative or fixed.

Relative datum axes

Relative datum axes reference curves, faces, edges, points, and other datums. You can create relative datum axes across multiple bodies.

Fixed datum axes

Fixed datum axes do not reference other geometry. Use any of the relative datum axis methods to create fixed datum axes by clearing the **Associative** box in the **Datum Axis** dialog box.

Where do I find it?

Application	Modeling
Toolbar	Feature® Datum/Point Drop-down®  Datum Axis
Menu	Insert® Datum/Point® Datum Axis
Graphics window	Right-click an object and choose Datum Axis

Datum axis types

Select an axis type from the **Type** option list.

When you edit a datum axis, you can change its type, defining objects, and associative status.



Inferred — Determines the best datum axis type to use based on objects you select.



XC-Axis — Creates a fixed datum axis on the XC-axis of the Work Coordinate System (WCS).



YC-Axis — Creates a fixed datum axis on the YC-axis of the WCS.



ZC-Axis — Creates a fixed datum axis on the ZC-axis of the WCS.



Point and Direction — Creates a datum axis from a specified point in a specified direction.



Two Points — Creates a datum axis by defining two points through which the axis passes.



On Curve Vector — Creates a datum axis tangent, normal, or binormal to a point on a curve or edge, or perpendicular or parallel to another object.



Intersection — Creates a datum axis at the intersection of two planar faces, datum planes, or planes.



Curve/Face Axis — Creates a datum axis on a linear curve or edge, or the axis of a cylindrical or conical face or torus.



Fixed — Available only when editing a datum axis.

Any datum axis created using the **YC-Axis**, **XC-Axis**, or **ZC-Axis**, or any of the other relative types used with the **Associative** check box cleared, appear as the **Fixed** type during an edit.

Datum axis options

 Reverse Direction	Cycle through the possible directions for the axis normal.
Associative	Clear this check box to create a fixed datum axis. ¹

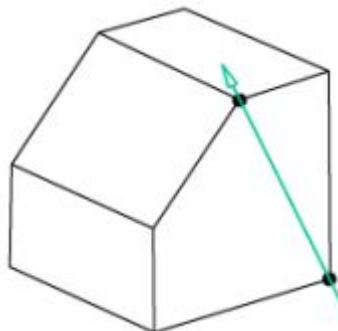
Applications for datum axes

- Define an axis of rotation for revolved features.
- Define an axis of rotation for circular arrays.
- Define a relative datum plane.
- Provide a directional reference.
- Use as a target for feature positioning dimensions.

1. In the **Part Navigator**, an associative datum plane has the name **Datum Axis**, while a non-associative datum plane has the name **Fixed Datum Axis**.

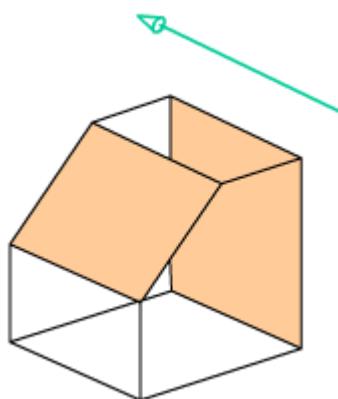
Create a datum axis through two points

1. On the **Feature Operation** toolbar, click **Datum Axis**  or choose **Insert→Datum/Point→Datum Axis**.
2. In the **Type** group, from the list, select **Two Points**.
3. Set the snap point options as desired.
4. Select two different point locations.
5. Click **OK**.



Create datum axis at an intersection

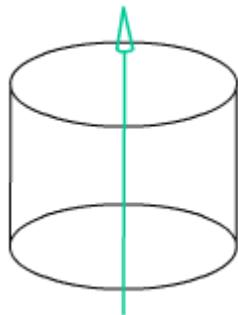
1. Click **Datum Axis** .
2. In the **Type** group, from the list, select **Intersection**.
3. Select the planar faces, datum planes, or planes.
4. Click **OK**.



Create a datum axis on a curve or face axis

1. Click **Datum Axis** 
2. In the **Type** group, from the list, select **Curve/Face Axis**.
3. Select the linear curve or edge, or the axis of a cylindrical or conical face or torus.
4. Click **OK**.

5





Datum CSYS overview

Use **Datum CSYS** to quickly create a coordinate system consisting of a set of **reference objects**. You can use the reference objects to **associatively** define the position and orientation of other features.

An existing datum CSYS at the absolute coordinate system origin is included in many of the default part templates.



A datum CSYS consists of the following reference objects:

- A coordinate system
- An origin point
- Three datum planes
- Three datum axes

5

You can create a datum CSYS:

- At a fixed location relative to the work or absolute coordinate systems.
- Associated to existing geometry.
- Offset from an existing datum CSYS.

Uses for reference objects in a datum CSYS

- Define placement faces, constraints, and position of sketches and features.
- Define vector directions for features.
- Define critical product locations in model space and control them with translation and rotation parameters.
- Define constraints to position parts in an assembly.

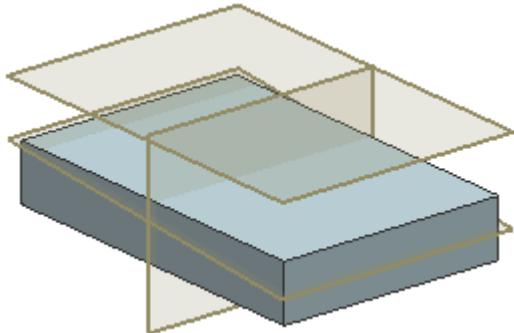
Where do I find it?

Application	Modeling
Toolbar	Feature® Datum/Point Drop-down® Datum CSYS
Menu	Insert→Datum/Point→Datum CSYS

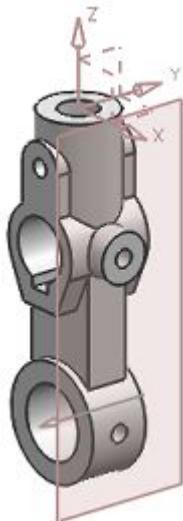
Activities: Datum features

In the *Datum features* section, do the activities:

- *Create relative datum planes*



- *Create datums on cylindrical faces*



Summary: Datum features

Datums are reference features that you use to help construct other features and sketches in locations and orientations where planar placement faces do not exist.

In this lesson you:

- Created associative datum planes and datum axes.
- Used datum features to help position other features.
- Edited datum planes to see how associative features are affected.
- Created an associative datum CSYS.

5

Lesson

6 *Swept features*

Purpose

This lesson introduces swept features that use a section string to define a solid or sheet body.

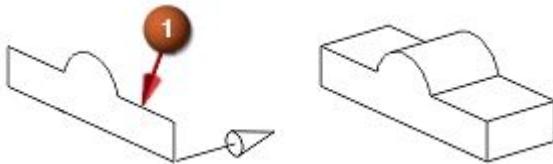
Objectives

- Create an Extrude feature.
- Create a Revolve feature.
- Create a Sweep Along Guide feature.
- Combine bodies using a Boolean operation.

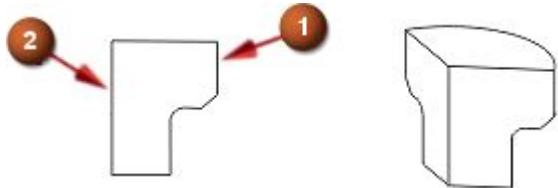
Types of swept features

You create swept features by extruding, revolving, or sweeping a section string. The section string may be composed of explicit curves, sketch curves, edges, or faces.

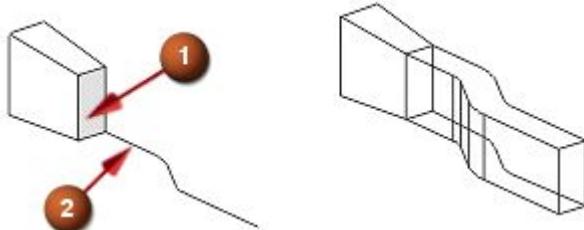
Extrude – Sweep a section string (1) in a linear direction for a specified distance.



Revolve – Rotate a section string (1) around a specified axis (2).



Sweep Along Guide – Sweep a section string (1) along a guide string (2).



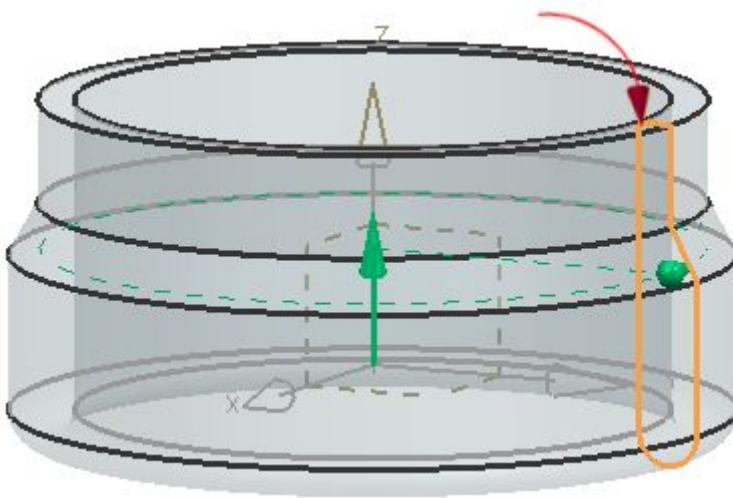
Swept bodies are associative with both the section string and the guide string.

Internal and external sketches

Sketches that you create from within NX commands like **Variational Sweep**, **Extrude**, or **Revolve** are *internal* sketches. The owning feature manages access to, and the display of, internal sketches. Use internal sketches when you want to associate the sketch with only one feature.



The sketch is only made visible when the owning feature, in this case a Revolved feature, is activated.



Sketches that you create independently using the **Sketch** command are *external* sketches, and are visible and accessible from anywhere within a part. Use an external sketch to keep the sketch visible and to use it in more than one feature or as reference for other sketches.

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Differences between internal and external sketches

- Internal sketches are visible in the graphics window only when you edit the owning feature.
- You cannot open an internal sketch directly from the Sketch task environment unless you first make the sketch an external feature.
- You can view external sketches in the graphics window and open them for editing without first opening the owning feature.

Internal and external sketch status change

Use this procedure to change the status of a sketch from internal to external and vice versa.

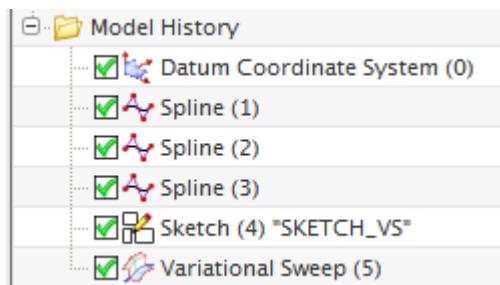
To change the status of a sketch from the **Part Navigator**.

- Right-click the owning feature and choose **Make Sketch External**.
- Right-click the owning feature and choose **Make Sketch Internal**.

The sketch is placed before its former owner in timestamp order.

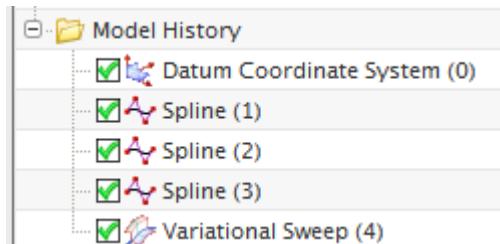


In the following navigator view, the external sketch is the fourth feature in the list, **SKETCH_VS**, and the owning feature, **Variational Sweep**, is fifth.



To reverse this operation, right-click the owning feature and choose **Make Sketch Internal**.

When you internalize the sketch, it no longer appears in the **Part Navigator**. Note that the **Variational Sweep** is now the fourth feature.

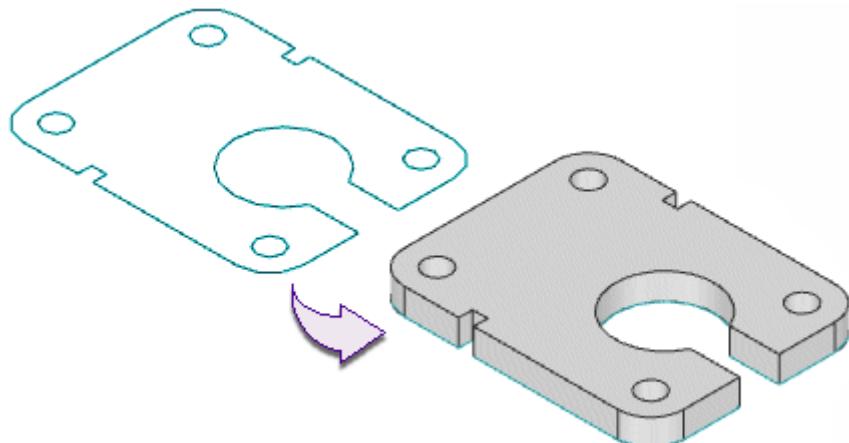




Extrude overview

Use the **Extrude** command to create a solid or sheet body by selecting a section of curves, edges, faces, sketches, or curve features and extending them a linear distance.

The following example shows how **Extrude** can form a solid body from a section of curves.



You can:

- Size an extrude feature by dragging distance handles or specifying distance values.
- Unite, subtract or intersect an extrude feature with existing bodies.
- Produce multiple sheet or solid bodies with a single extrude feature.
- Trim an extrude feature using faces, datum planes or solid bodies.
- Add drafts to an extrude feature.
- Add offsets to an extrude feature, measured from its base section.

6

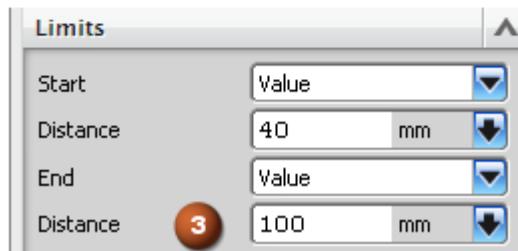
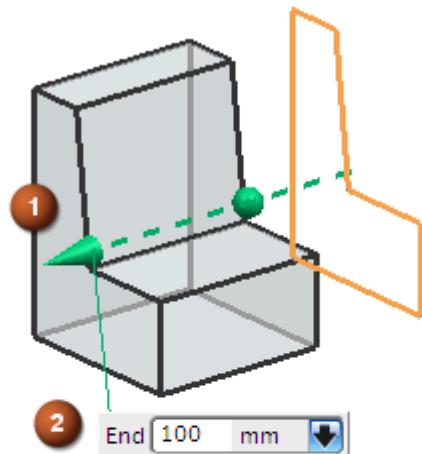
Where do I find it?

Application	Modeling
Toolbar	Feature® Extrude
Menu	Insert® Design Feature® Extrude
Shortcut menu	Right-click sketch® Extrude

Extrude start and end distances

To specify the start and end distances of an extrude feature:

- Use drag handles (1).
- Specify values in on-screen input boxes (2).
- Specify values in dialog boxes (3).



Create a simple extruded feature

1. Click **Extrude** 

2. Select a sketch, curves, or edges for the section.

Selection Intent is available.

The default direction of the extrude is normal to the plane of the section.

3. Specify **Start** and **End** limits by using the drag handles in the graphics window or typing distance values.

4. Select a **Boolean** type.

To create a new solid body, select **None**.

To combine the feature with an existing solid body, select one of the other Boolean types.

5. Click **Apply** or **OK** to create the extrude feature.

Combining bodies using Boolean commands

Use Boolean commands to combine solid bodies or sheet bodies.

-  **Unite** – Combines the volume of two or more solid bodies into a single body.

The target and tools must overlap or share faces so that the result is a valid solid body.

-  **Subtract** – Removes the volume or area of one or more tool bodies from a target body.

The target and tools can be solid bodies or sheet bodies.

-  **Intersect** – Creates a body containing the shared volume or area between a target body and one or more tool bodies.

You can intersect solids with solids, sheets with sheets, and sheets with solids. If you intersect a sheet and a solid, the sheet body must be the target.

Target and tool

When you use a Boolean command, you identify a target body and one or more tool bodies.

- The target body is modified by the tool bodies.
- You can save unmodified copies of the target and tool bodies.

Where do I find it?

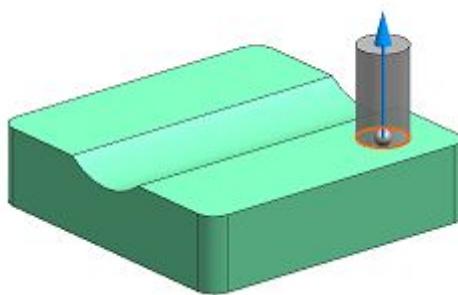
Application	Modeling
Toolbar	Feature® Combine Drop-down list® Unite  Subtract   Intersect
Menu	Insert® Combine® Unite/Subtract/Intersect
Location in dialog box	Boolean group in some dialog boxes to combine features with an existing body as you create them.



Extrude – Inferred Boolean

The **Extrude** command automatically infers the Boolean operation that you may want to apply. **Inferred** is the default Boolean option.

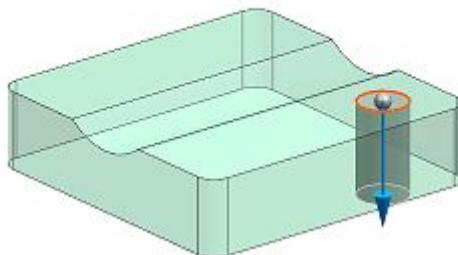
- The inferred Boolean operation is based on the extrude direction and section normal. The tool body must touch or intersect the target body for a Boolean to be inferred.



Case 1

Direction is away from target; most logical inferred Boolean is Unite.

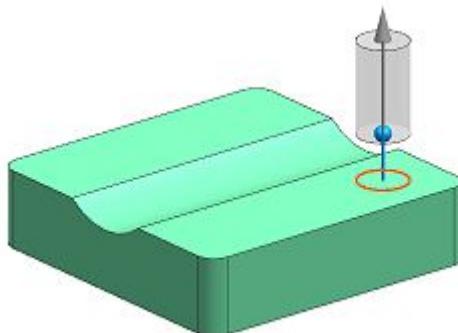
Inferred Boolean = Unite
Status Line: <i>Boolean will be a Unite</i>



Case 2

Direction is into target; most logical inferred Boolean is Subtract.

Inferred Boolean = Subtract
Status Line: <i>Boolean will be a Subtract</i>



Case 3

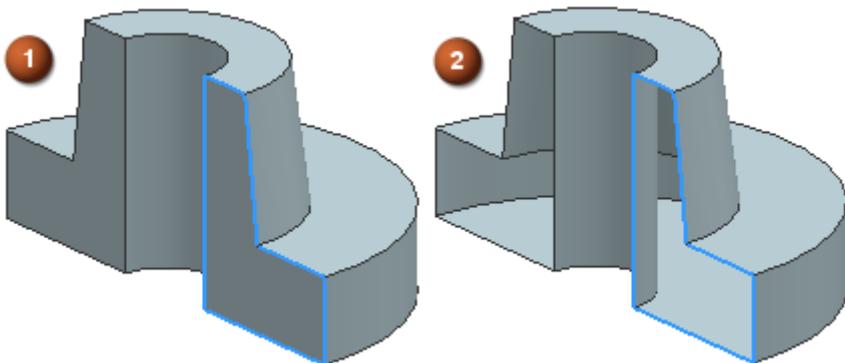
No possible Boolean; none inferred.

Inferred Boolean = none
Status Line: <i>No Boolean will be performed</i>

- The **Inferred** option is available in the **Boolean** group of the **Extrude** dialog box and is the default.
 - If the inferred Boolean operation is not suitable, you must directly select a feasible Boolean operation.
 - After you specify a Boolean type, that type becomes the default Boolean operation the next time you use **Extrude**.
 - When an **Extrude** feature is edited, the actual Boolean used (even if inferred) will appear in the Extrude dialog box instead of Inferred.
- When possible, the **Extrude** command will also infer a target body. You can change this inferred target by selecting a different target.

Body type

You can use the **Extrude** and **Revolve** commands to create a solid body (1) or a sheet body (2).



You create a solid body when you specify:

- a closed section with the **Body Type** option set to **Solid**.
- an open section with an offset.
- an open section and revolve a total angle of 360°.

You create a sheet body when you specify:

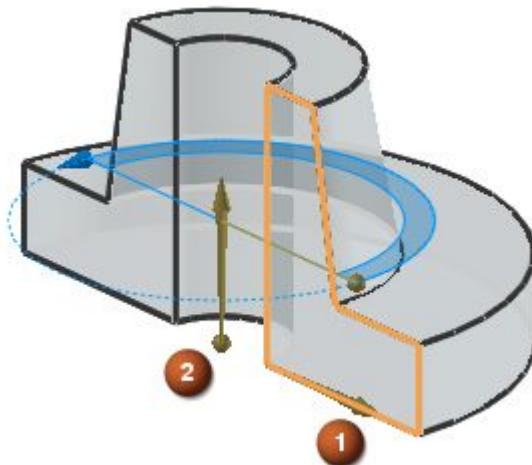
- a closed section with the **Body Type** option set to **Sheet**.
- an open section with no offset. For revolve, the total angle must be less than 360°.



Revolve overview

Use this command to create a round or partially round feature by rotating section curves around an axis.

The following graphic shows a section ① rotated around an axis ② from 0 to 180 degrees.



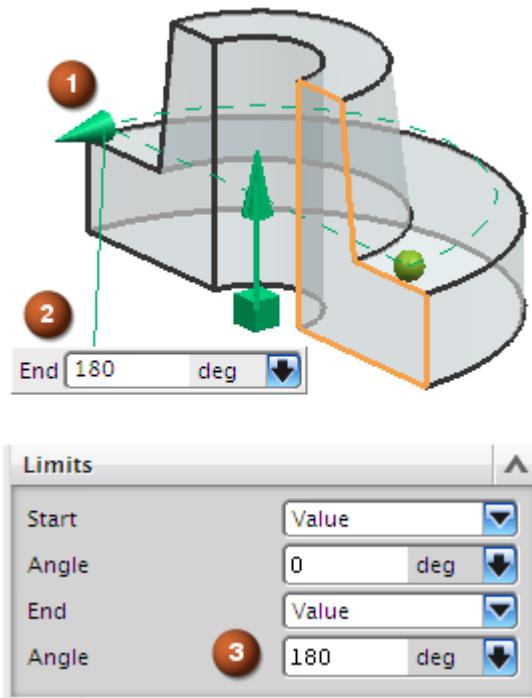
Where do I find it?

Application	Modeling
Toolbar	Feature® Revolve
Menu	Insert® Design Feature® Revolve
Shortcut menu	Right-click sketch® Revolve

Revolve start and end angles

To specify the start and end angles of a revolve feature:

- Use drag handles (1).
- Specify values in on-screen input boxes (2).
- Specify values in dialog boxes (3).



Specifying vectors using the OrientXpress tool

Use the **OrientXpress** tool to quickly identify a principle axis.



The tool works in conjunction with NX commands that require an orientation input. It will appear when you specify a vector for a revolve axis or an extrude direction. You can select an axis on the tool to specify a vector.

You can move the tool by clicking one of the arrowheads and dragging it to a new location.

Create a simple revolved feature

1. Click **Revolve** 

2. Select a sketch, curves, or edges for the section.

Selection Intent is available.

3. Click the middle mouse button or click **Specify Vector** in the **Axis** group in the dialog box.

4. Specify a rotation axis through one of the following:

- In the graphics window, select a curve, edge, relative datum axis or plane about which to rotate the section.
- Define a rotation axis using vector methods or the **Vector Constructor** in the Axis group in the dialog box. If the vector you specify does not have an implied point, you may need to define one using **Specify Point** or the **Point Constructor**.

5. Specify **Start** and **End** limits by using the drag handles in the graphics window or typing angle values.

6. Select a **Boolean** type.

To create a new solid body, select **None**.

To combine the feature with an existing solid body, select one of the other Boolean types.

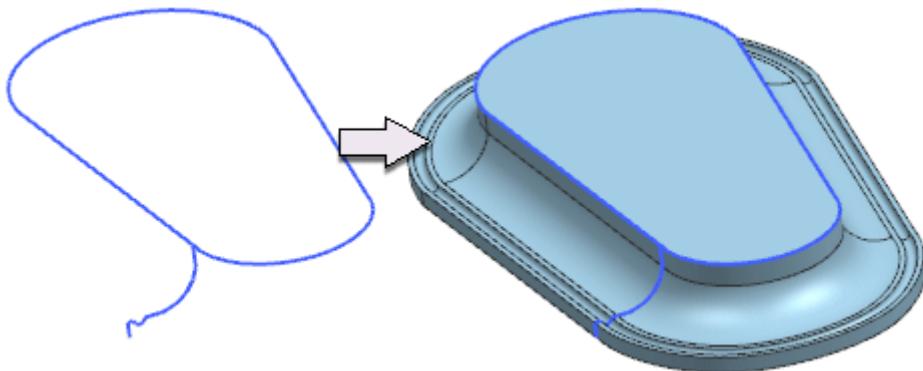
7. Click **Apply** or **OK** to create the feature.



Sweep along Guide overview

Use the **Sweep along Guide** command to create a body by sweeping one section along one guide. You can:

- Select a section and a guide consisting of connected sketches, curves, or edges.
- Select a guide that contains sharp corners.
- Create a solid body or a sheet body.



If you want to select multiple sections, multiple guides, or control the interpolation, scale, and orientation of the sweep, use the **Swept** command.

6

Where do I find it?

Application	Modeling
Toolbar	Surface® Sweep Drop-down® Sweep along Guide
Menu	Insert® Sweep® Sweep along Guide

Create a simple sweep along guide feature

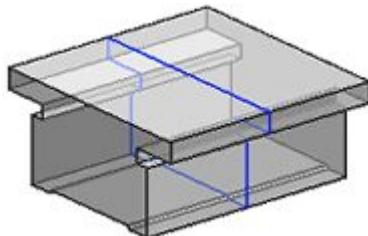


1. Click **Sweep along Guide**.
2. Select section curves or edges.
Selection Intent is available.
3. Click the middle mouse button or click **Select Curve** in the **Guide** group in the dialog box.
4. Select guide curves or edges.
Selection Intent is available.
5. Select a **Boolean** type.
To create a new solid body, select **None**.
To combine the feature with an existing solid body, select one of the other Boolean types.
6. Click **Apply** or **OK** to create the feature.

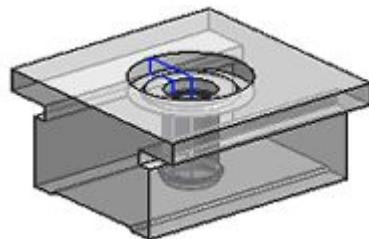
Activities: Swept features

In the *Swept features and Boolean operations* section, do the activities:

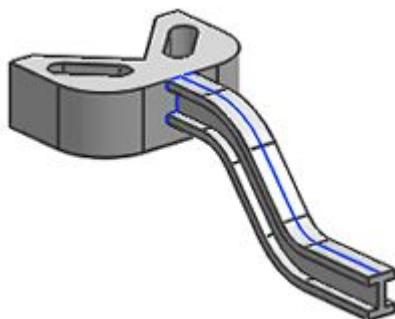
- *Extrude a sketch*



- *Revolve a sketch*



- *Sweep along an open guide*



Summary: Swept features

Use swept features to define solid or sheet bodies using a section. A section can be a sketch, or a collection of curves and face edges.

In this lesson you:

- Extruded a sketch.
- Revolved a sketch.
- Combined bodies using a Boolean operation.
- Swept sections along open and closed guide strings.

Lesson

7 *Swept feature options*

Purpose

This lesson introduces draft, offsets, and selection intent to define profiles and swept features.

Objectives

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

- Apply selection intent to define sections from intersecting curves and multiple loops.
- Create an extruded feature with offsets.
- Create an extruded feature with draft.

Selection Intent - Curve Rule

The Selection bar has rules you can set up when selecting curves for sweep operations.



1. **Curve Rule** options

2. **Curve Rule** modifiers

Use these rules to help you to select curves or edges:

- In fewer steps than selecting them individually.
- When only part of some curves is needed.
- When a rule can determine which branch to take at multi-curve intersections.
- When future model development or edits may change the number of curves in the profile.

Curve Rule options

Single Curve	Individually select one or more curves or edges with no rule.
Connected Curves	Select a chain of curves or edges that share endpoints. This rule does not grow or shrink the chain if curves are added or no longer form a single chain after an edit to the model.
Tangent Curves	Select a tangent chain of curves or edges. No rule is applied if the chained curves are non-associative. Non-associative curves that are no longer tangent after an edit are not discarded.
Feature Curves	Collect all output curves from curve features, such as sketches or any other curve features.
Face Edges	Collect all edges of the face containing the edge you select. When you select an edge, the cursor location determines which face is selected.
Sheet Edges	Collect all edges of the sheet body you select.
Region Boundary Curves	Select a profile that encloses an area with a single mouse click.
Curves in Group	Selects all curves that are members of a selected group.
Infer Curves	Use the default intent method for the type of object you select. For example, with Extrude the default is Feature Curves if you select a curve, and Single if you select an edge.

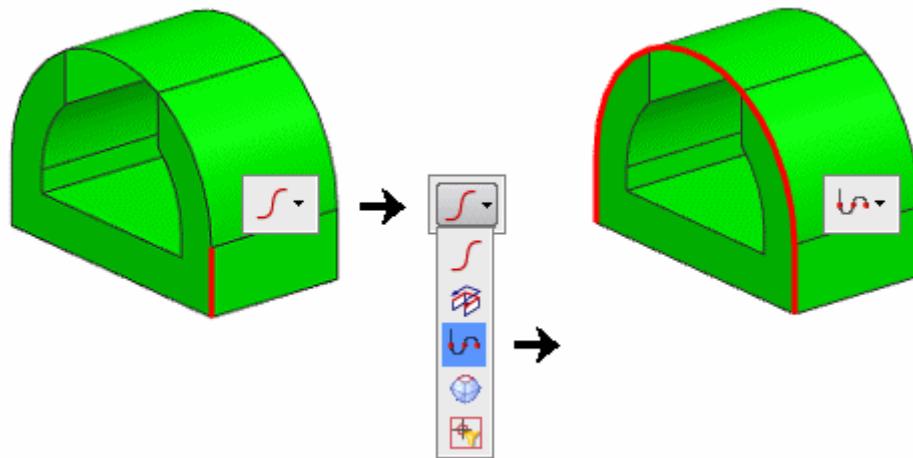
Active Selection mini toolbar

The *Active Selection mini toolbar* appears above your cursor when you first select a curve or edge. You can select a different Selection Intent curve rule with minimal cursor movement, while remaining focused on the geometry.



The Active Selection mini toolbar only appears when you select a curve or edge when the current Selection Intent rule is one of the following:

- **Infer Curves**
- **Single Curve**



Changing the Single Curve Selection Intent rule to Tangent Curves



Options appear on the toolbar with both icons and text, but the above figure shows only the icons.

7

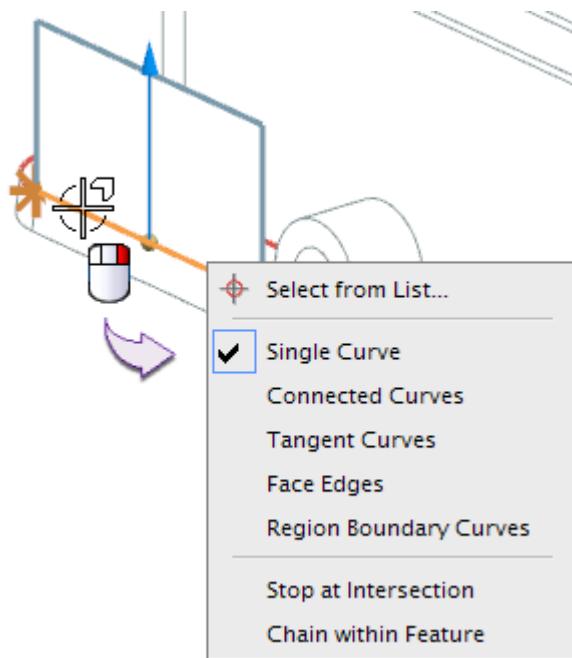
You can:

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

Changing selection intent rules

The Active Selection mini toolbar is used to change a selection intent rule *immediately* after an object is first selected.

If you want to change the rule in effect after the initial object selection, you can right-click over the object and select a different rule from the menu.



Curve collection modifiers

	Stop at Intersection	Specify that auto chaining stops on intersection points of wireframe.
	Follow Fillet	Automatically follow and leave fillets or circular curves during section building.
	Chain within Feature	Limit the chaining to collect curves only from the parent feature of the selected curve.
	Path Selection	Define a continuous path in a complex network of curves and edges with a minimal number of selections. Path Selection is available for all commands that are section-based.

Extrude start and end limits

Use the limit options to define the overall construction method and the extents of the extrude feature.

Options

Value	Specify numeric values for the start or end of the extrusion.
Until Next	Extends the extrude feature to the next body along the direction path.
Until Selected	Extends the extrude feature to a face, datum plane, or body that you select.
Until Extended	Trims the extrude feature (if it is a body) to a face you select when the section extends beyond its edges.
Symmetric Value	Converts the Start limit distance to the same value as the End limit.
Through All	Extends the extrude feature completely through all selectable bodies along the path of the specified direction.

Extrude with offset

The **Offset** options lets you specify up to two offsets to the profile for extruded and revolved sections. You can assign unique values for both offsets.

You can:

- Type values for the offsets in the **Start** and **End** boxes in the dialog box.
- Type values in on-screen input boxes in the graphics window.
- Drag the offset handles.

Options

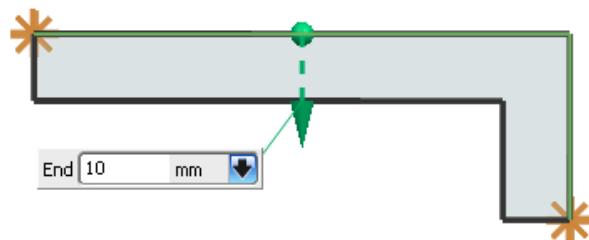
None	Create no offset.
Single-Sided	Add a single end offset to the extrude.
Two-Sided	Add an offset with duplicate start and end values, measured from opposite sides of the section. The value for both start and end is determined by the last one you specify.
Symmetric	Add an offset with duplicate start and end values.
Start	Start the offset at the value you specify, measured from the section.
End	End the offset at the value you specify, measured from the section.

Two sided offset examples

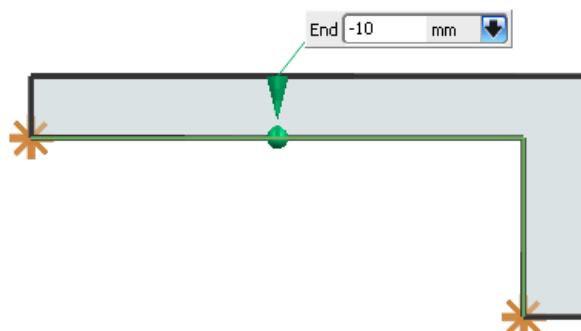
The start and end offset values may be positive or negative.

The positive direction is shown by the **End Offset** drag handle.

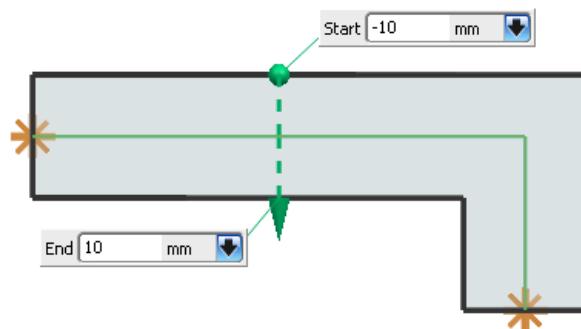
Start Offset Zero, End Offset Positive



Start Offset Zero, End Offset Negative



Start Offset Negative, End Offset Positive



Extrude with draft

Use the **Draft** option to add a slope to one or more sides of the extrude feature, in one or two directions from the section.



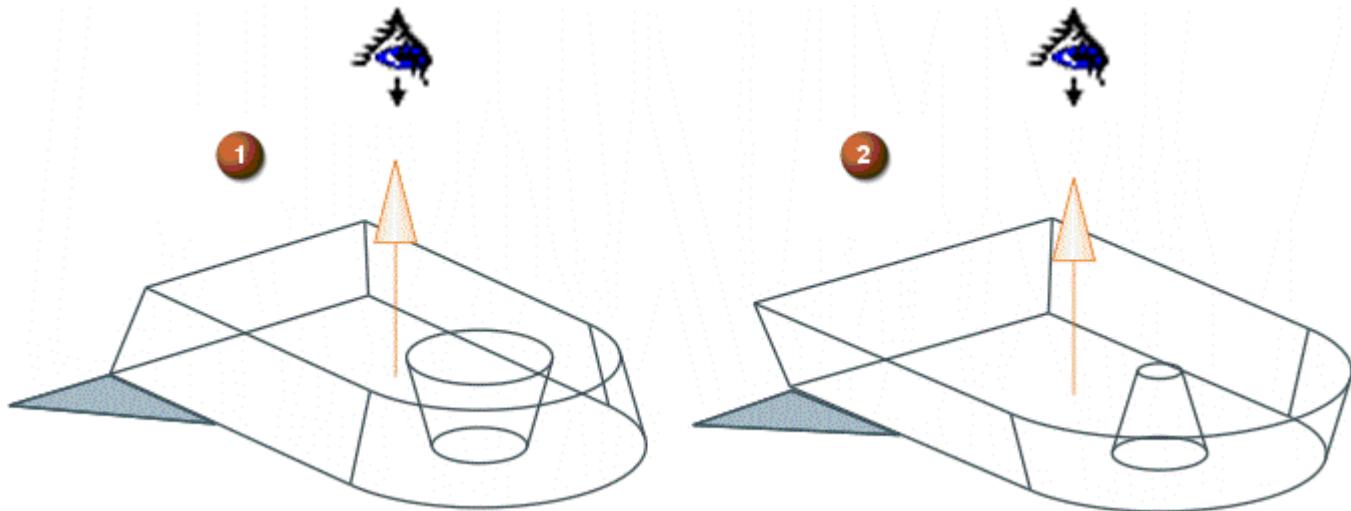
You can apply a draft only when the extruded section is *planar*.

Option	Description
None	No draft is created.
From Start Limit	Maintain the original size of the extruded section at the start limit.
From Section	Maintain the original size of the extruded section at the section plane.
From Section-Asymmetric Angle	Split the side faces into two sides at the section plane. You can control the draft angle separately on each side of the section. ¹ Front Angle and Back Angle options appear; one pair with the Single option, and one pair for each set of tangent curves for the Multiple option.
From Section-Symmetric Angle	Split side faces at the section plane, and use the same draft angle on both sides. ¹
From Section-Matched Ends	Maintain the original size of the extruded section, and split the side faces of the extrude feature at the section plane.
Angle Option	Match the size of the shape at the end limit to that of the start limit, and vary the draft angle to maintain the matched shape at the end limit. ¹ Single — Specify a single draft angle for all faces of the extrude feature. Multiple — Specify unique draft angles to each tangent chain of faces of the extrude feature.
Angle List	Specify a value for a draft angle. Examine the name and value for each draft angle. The list appears when the Angle Option is set to Multiple .

1. Available only when the extrude extends from both sides of the section.

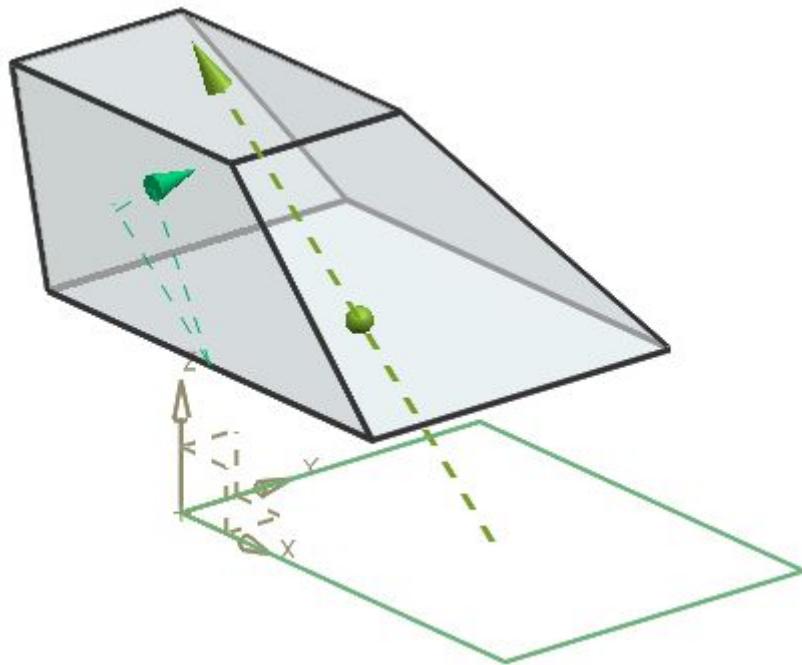
Positive and negative draft angles

If you look at the body with your eye positioned with respect to the draft vector as shown, positive draft angles (1) enable you to see the draft feature faces, and negative draft angles (2) hide the draft feature faces.



Draft and the extrude direction

Draft is measured with respect to the extrude direction. The extrude direction does not need to be perpendicular to a planar section.



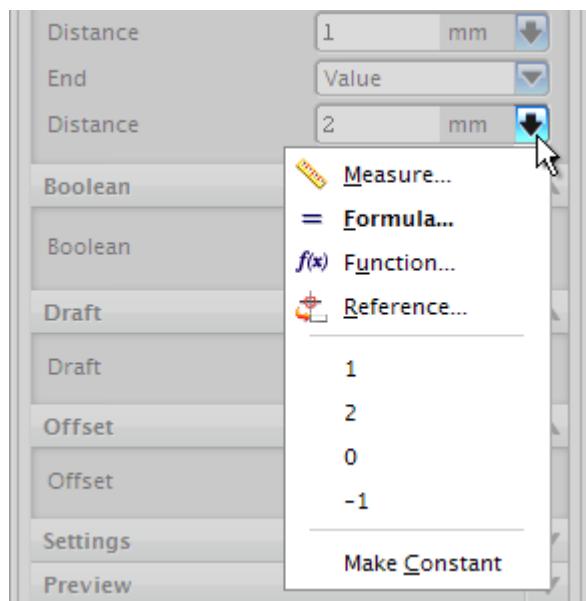
DesignLogic parameter entry options

Parameter entry options let you define your model parametrically as you specify feature values.

To access the options, click  next to the text input box.

You may specify a value based on a:

- Measurement.
- Formula.
- Math or knowledge-based function.
- Reference to an existing value.
- Conversion of the above to a constant.
- Recently used value that you want to use again.



Reference existing parameters

1. From the parameter entry options list, select **Reference**.

The **Parameter Selection** dialog box appears.

Initially, the list is empty.

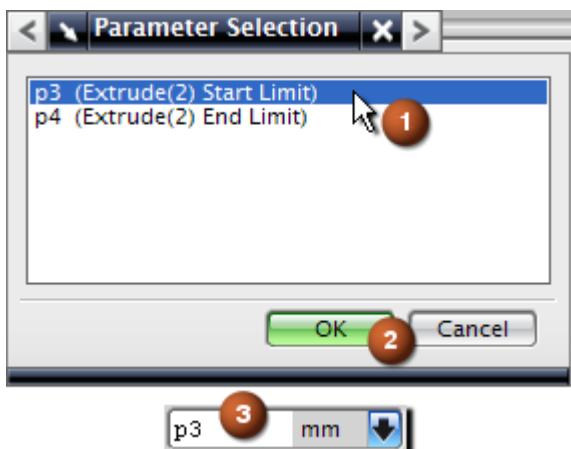
2. Select an existing feature.

The list is populated with the feature's parameters and their descriptions.

3. Select a parameter (1).

4. Click **OK** (2).

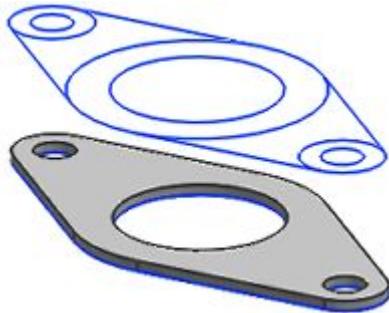
The parameter name now appears in the box (3).



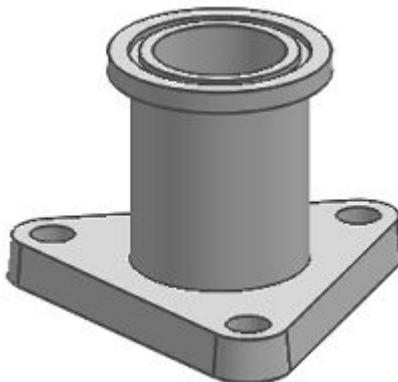
Activities: Swept feature options

In the *Swept feature options* section, do the activities:

- *Extrude using selection intent*



- *Extrude with offsets*



Summary: Swept feature options

Use selection intent to quickly specify sections by applying rules to complex sets of curves.

Extrude or revolve with offsets to thicken simple sections or alter sections.

Incorporate draft in extruded features instead of using separate draft features to simplify your history tree.

Use **DesignLogic** to increase productivity when modelling parametrically.

In this lesson you:

- Applied selection intent to define sections.
- Extruded with offsets.
- Extruded with draft.

Lesson

8 Trim Body

Purpose

This lesson introduces the **Trim Body** command to define the topology of a solid body.

Objectives

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

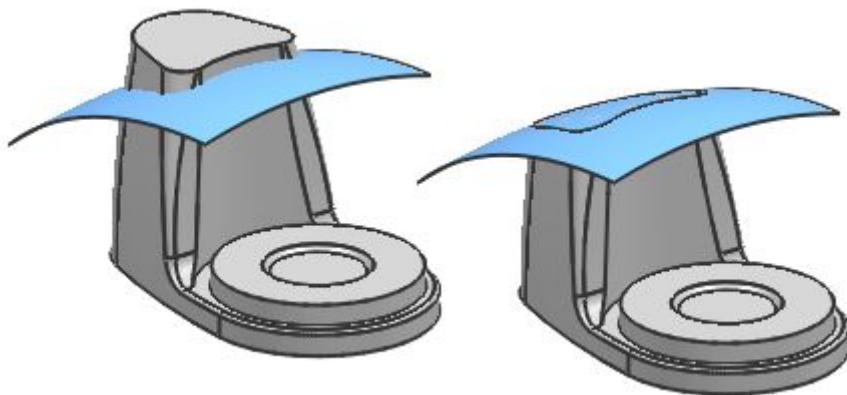
- Trim a solid body to a plane or sheet body.



Trim Body overview

Use **Trim Body** to trim one or more target bodies using a face or plane. You can specify the portion of the body to retain and the portion to discard. The target bodies take the shape of the trimming geometry.

- You must select at least one target body.
- You can select a single face, multiple faces from the same body, or a datum plane to trim the target bodies.
- You can define a new plane to trim the target bodies.



Where do I find it?

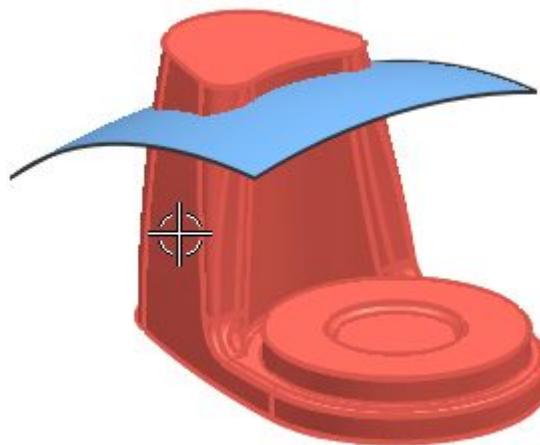
Application	Modeling
Toolbar	Feature® Trim Body
Menu	Insert® Trim® Trim Body



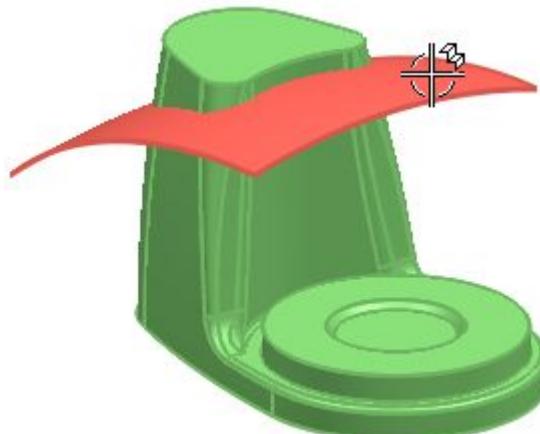
Trim a solid body to a face

This example shows how to trim a single, solid target body to the face of a sheet body.

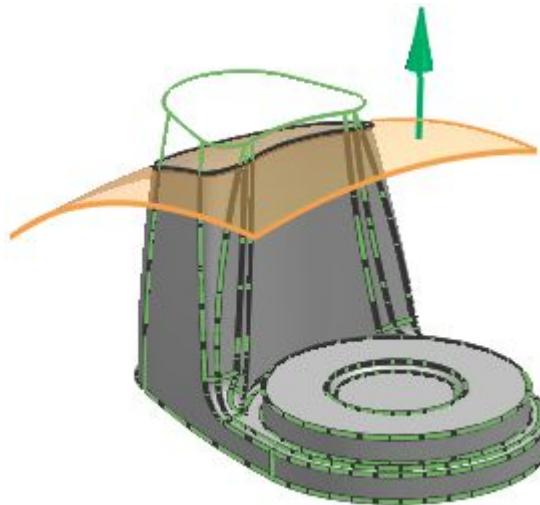
1. On the **Feature** toolbar, click **Trim Body** or choose **Insert® Trim® Trim Body**.
2. Select the target body to trim.



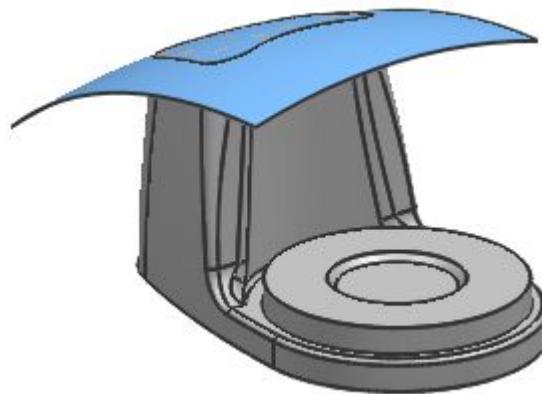
3. In the **Tool Option** list, make sure **Face or Plane** is selected.
4. In the **Tool** group, make sure that **Select Face or Plane** is active.
5. Select the face of the sheet body.



A vector points toward the portion of the target body to be removed.



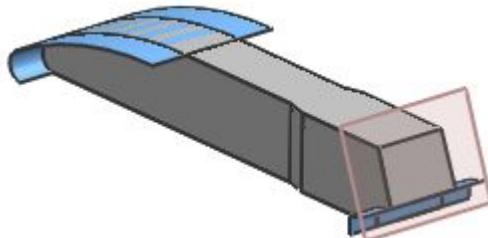
6. (Optional) If the vector is not pointing toward the portion of the target body you want to remove, click **Reverse Direction** .
7. Click **OK** or **Apply** to create the Trim Body feature.



Activities: Trim Body

In the *Trim Body* section, do the activity:

- *Trim a solid body*



Summary: Trim Body

Use the **Trim Body** command to define the topology of a body.

In this lesson you:

- Trimmed a solid body to a plane and a sheet body.

Lesson

9 Hole features

Purpose

This lesson introduces the general hole feature.

Objectives

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

- Create general hole features.
- Position hole features.
- Edit the parameters and location of hole features.



Hole overview

Use the **Hole** command to add the following types of hole features to one or more solid bodies in a part or assembly:

- General holes (simple, counterbored, countersunk, or tapered form)
- Drill size holes
- Screw clearance holes (simple, counterbored, or countersunk form)
- Threaded holes
- Hole Series (series of multi-form, multi-target body, aligned holes in the work part or the assembly)

You can:

- Create holes on non-planar surfaces.
- Create multiple holes by specifying multiple placement points.
- Specify the position of holes by sketching. You can use the Snap Point and Selection Intent options to select existing points or feature points.
- Create holes using formatted data tables for the Screw Clearance Hole, Drill Size Hole, and Threaded Hole types.
- Use standards like ANSI, ISO, DIN, JIS and so on.
- Use the **None** and **Subtract** Boolean commands on the target bodies while creating a Hole feature.
- Optionally add start, end, or relief chamfers to the hole feature.

Where do I find it?

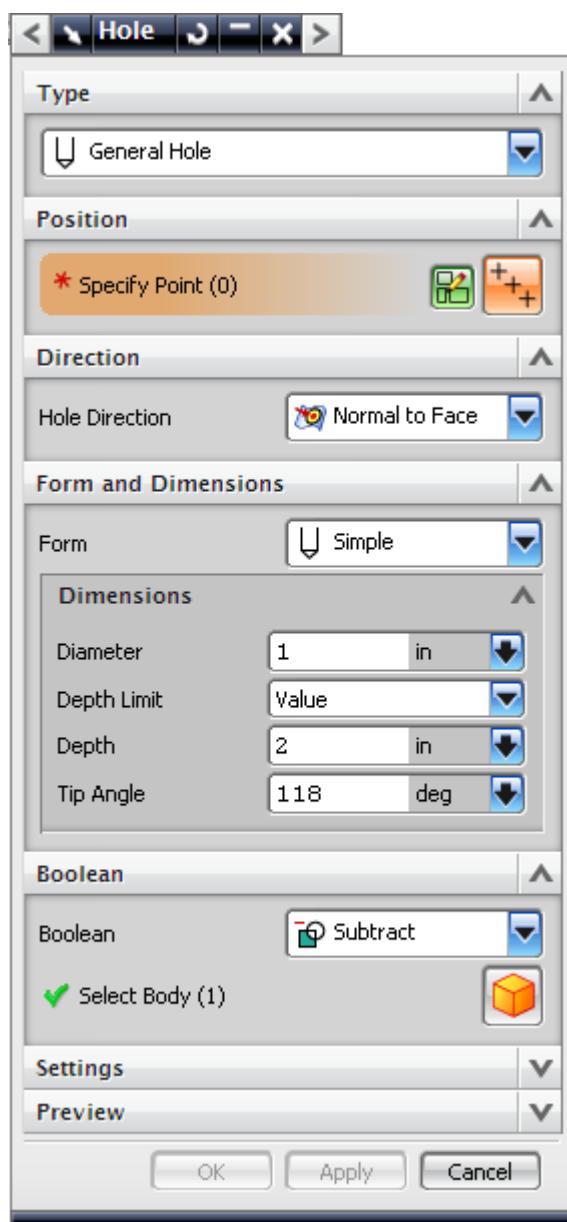
Application	Modeling
Toolbar	Feature→Hole
Menu	Insert→Design Feature→Hole

Hole dialog box

The **Hole** dialog box includes four groups that are unique to hole features.

- Type
- Position
- Direction
- Form and Dimensions

The options available within the groups will change depending on which type and form you select.



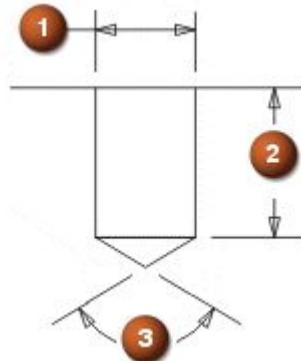
Hole position and direction options

Position
Specify Point
Direction
Hole Direction
<p>Specifies the position of the Hole feature.</p> <p>You can use one of the following methods to specify the center of the hole:</p> <ul style="list-style-type: none"> • Sketch Section  — Create a sketch to define the location of the center of the hole. • Point  — Specify the center of the hole using existing points. Use Snap Point and Selection Intent options to select existing points or feature points.
<p>Specifies the hole direction. The default hole direction is along the - ZC axis. You can define the hole direction using one of the following options:</p> <ul style="list-style-type: none"> • Normal to Face — Defines the direction of the hole along the direction opposite to the face normal which is nearest to each of the specified points. • Along Vector — Defines the hole direction along the specified vector.

Hole form and dimension options

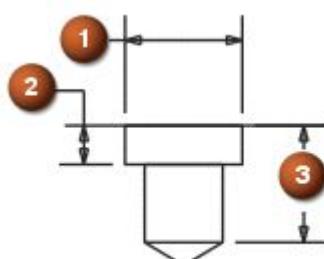
Simple

1. Diameter
2. Depth
3. Tip Angle



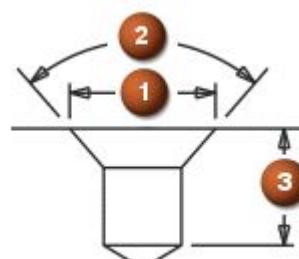
Counterbore

1. C-Bore Diameter
2. C-Bore Depth
3. Depth



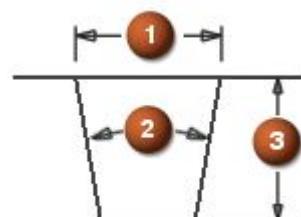
Countersink

1. C-Sink Diameter
2. C-Sink Angle
3. Depth



Tapered

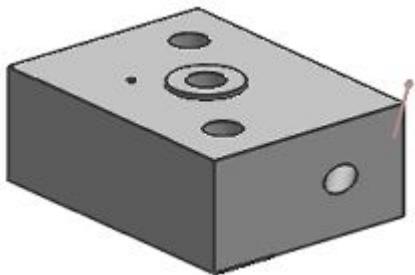
1. Diameter
2. Taper Angle
3. Depth



Activities: Hole features

In the *Hole features* section, do the activities:

- *Create general holes*



Summary: Hole features

Hole features are one of several features with predefined shapes.

In this lesson you:

- Created general hole features.
- Positioned a hole using a sketch.
- Edited parameters and locations of holes.

Lesson

10 Expressions

Purpose

This lesson introduces expressions which define characteristics of features.

You can easily create many types of intelligent expressions based on measurements and inter-part references.

Objectives

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

- Create expressions.
- Edit expressions.
- Create conditional expressions
- Reference measurements of geometric properties via expressions

Expressions overview

Expressions are arithmetic or conditional formulas that define the characteristics of features.

Software expressions are automatically created when you:

- Create a feature.
- Dimension a sketch.
- Position a feature.
- Constrain an assembly.

All expressions have a unique name and a formula that can contain a combination of variables, functions, numbers, operators, and symbols.

Expression names are variables that you can insert in the formula strings of other expressions. This can be used to break up lengthy formulas and to define relationships that can be used in place of numbers.

Where do I find it?

Menu	Tools→Expression
Shortcut menu	Right-click an expression in the Part Navigator , either in the Main panel or Details panel, and choose Edit in Expression Editor From supported Modeling dialogs, click parameter entry options and choose Formula .

Expression examples

Here are some examples of expressions, their formulas and their resulting values:

Expression Name	Formula	Value
length	5*width	20
p39 (Extrude(6) End Limit)	p1+p2*(2+p8*sin(p3))	18.849555921
p26 (Simple Hole(9) Tip Angle)	118	118

Expressions case sensitivity

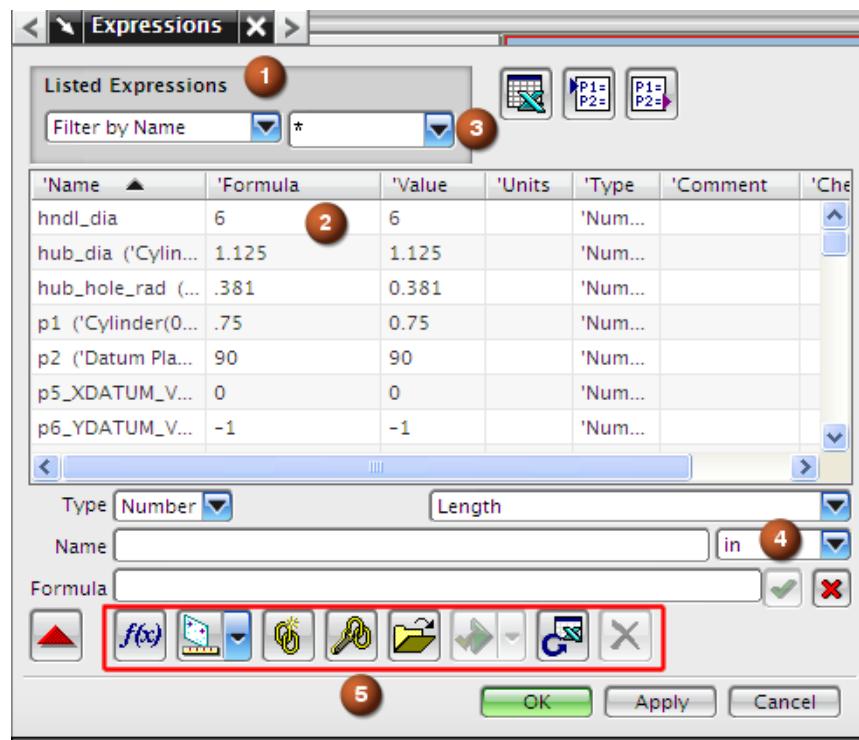
Expression names are *not* case sensitive, with the following *exceptions*:

- Expression names are case sensitive if their dimensionality is set to **Constant**.
- Expression names are case sensitive if they were created before NX 3.

When expression names are case sensitive, the name must be spelled *exactly* when used in other expressions.

The Expressions dialog box

1. **Listed Expressions** filter the expression list by either name, value, formula, measurements, unused expressions, object parameters or all expressions.
2. **Expression list** lists all expression based on the type of filter used. The list columns provide additional information such as name, value, formula, or mathematical equation.
3. Filter box lets you enter additional filtering strings. The string is evaluated based on the filter category type (that is, **Filter by Name**, **Filter by Value**, or **Filter by Formula**)
4. Unit is available only when the **Type** is set to **Number** and the Dimensionality to something other than **Constant**. Specifies the unit for the selected dimensionality. If you change the dimensionality type, the unit also changes.
5. Additional functions are Functions, Measurements, Create Interpart Reference, Edit Interpart Reference, Open Referenced Parts, Requirements, Refresh Values from External Spreadsheet, Delete.



Creating expressions

- Software generated expressions are created for you automatically when you create features and dimension sketches. They have the format p#, where # represents a series of whole numbers.
- Enter user defined expressions in an on-screen input box as the name and formula, separated by the equal symbol, for example **Rad=5.00**.
- In the **Expressions** dialog box, enter an expression name in the **Name** box and the corresponding formula in the **Formula** box.



After you type the name of the expression, you may press the Tab, equal sign, or Enter key to advance the cursor to the **Formula** box, or just click in the **Formula** box.

Create a numerical expression

1. Choose **Tools→Expression**.
2. In the **Name** box, type the name of the expression and press Enter.
3. (Optional) Change the default values in the **Dimensionality** and **Units** lists.
4. In the **Formula** box, type the formula for the expression and press Enter.
5. Click **Apply** or **OK** to save expression changes.

Edit an expression

1. Choose **Tools→Expression**.
2. Select an expression to modify from the list.
3. Modify the **Name**, **Formula**, **Dimensionality**, or **Units** of the expression.



For *software generated expressions*, **Dimensionality** and **Units** are *not* available for editing.

4. Press Enter or **Accept Edit**
5. Click **Apply** or **OK** to save the expression.



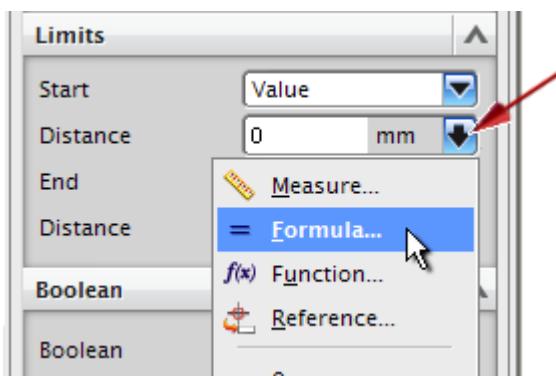
When you edit the name of an expression, NX automatically updates the formula of any expression that references it.

Parameter entry options

Access the **Expressions** dialog box as you create features by choosing **Formula** from the parameter entry option menu.

You can specify a formula for the expression referenced by a feature parameter.

Parameter entry options are available with most text input boxes.



Inserting functions into a formula

Use the **Function**  option to search for a function and insert it into a formula.

Examples of the provided functions include the following:

abs	Returns the absolute value of a given number.
arcsin	Returns the inverse sine of a given number in degrees
sin	Returns the sine of a given number with units defined as constant, degrees, or radians.
sqrt	Returns the square root of a given positive number.
pi	Returns the value of pi. This function requires no input arguments.



The system will handle unit conversions automatically if, for example, you specify inches in a metric part

Listing expressions associated with features

If **Listed Expressions** is set to **All**, all of the expressions in the part are listed.

If **Listed Expressions** is set to **Object Parameters**, and a feature is selected, only the expressions associated with the selected feature are listed.

If an expression defines a feature, the feature name is listed with it. For example, **p8 (Simple Hole(5) Diameter)**

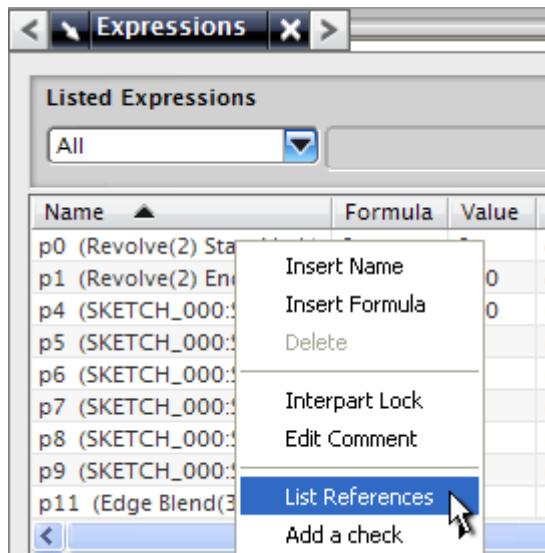
You can list all of the expressions associated with a feature in an Information window:

- Choose **Information→Feature** and select the feature.
- Over a feature node in the **Part Navigator**, choose **Information** from the shortcut menu.

List Referencers

The **List Referencers** command shows you if an expression is referenced in another expression and what feature(s) use the expression.

Over a listed expression, from the shortcut menu, choose **List Referencers**.



Insert Name

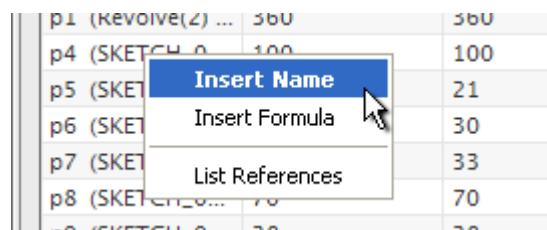
The **Insert Name** option places the name of a selected expression into a formula you are editing.

Over a listed expression, from the shortcut menu, choose **Insert Name**.



Bold type on an option in the shortcut menu for an object, **Insert Name** for an expression name, for example, means that the option in bold type is performed when you double-click the object.

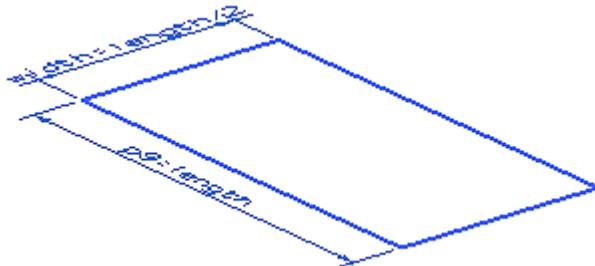
When you are editing a formula, you can double-click a listed expression to insert its name.



Activities: Expressions

In the *Expressions* section, do the activity:

- *Create and edit expressions*



Conditional expressions

Expressions can be used to define a variable based on specific conditions. This kind of expression is created by using the if-else statement.

Consider the following:

Example	
Name	Formula
Lgth	12.5
Wdth	if(Lgth > 10) (5) else (3)

The Expression for Wdth is defined depending on the following statement:

- If Lgth is greater than 10, Wdth is equal to 5.
- If Lgth is less than or equal to 10, Wdth is equal to 3.

Expressions can also use Boolean operations such as AND or OR.

Consider the following:

Example	
Name	Formula
Lgth	12.5
Wdth	if(Lgth > 0 && Lgth < 10) (3) else (5)

Here, the Expression for Wdth is defined depending on the following statement:

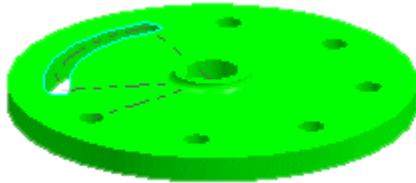
- If Lgth is greater than 0 AND less than 10, the value for Wdth is 3.
- Otherwise the value of Wdth is 5.

Syntax and the command portions of the statement must be lower case.

Activities: Create conditional expressions

In the *Expressions* section, do the activity:

- *Create conditional expressions*



Summary: Expressions

Expressions are algebraic or arithmetic formulas that define the characteristics of features.

In this lesson you:

- Created expressions.
- Edited expressions.
- Created conditional expressions.
- Created measurement expressions.

Lesson

11 Coordinate systems

Purpose

This lesson introduces the coordinate systems that are used in NX.

Objectives

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

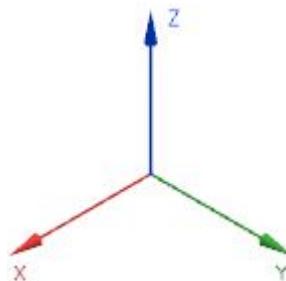
- Describe the differences between the absolute coordinate system (ABS) and the work coordinate system (WCS).
- Move the WCS.
- Obtain geometry information relative to the WCS.

Coordinate systems overview

There are different coordinate systems in NX. A three-axis symbol is used to identify a coordinate system.

The intersection of the axes is called the *origin* of the coordinate system.

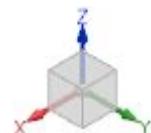
The coordinate values of the origin are X = 0, Y = 0, and Z = 0. Each axis line represents the positive direction for that axis.



You can define planes and coordinate systems for constructing geometry. These planes are completely independent of the viewing direction.

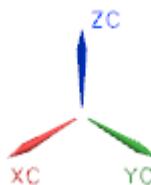
The most frequently used coordinate systems for design and model creation are:

- Absolute Coordinate System



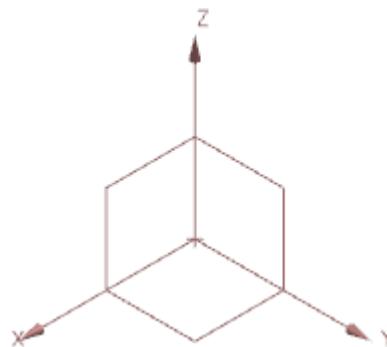
There is one absolute coordinate system. It is not visible or movable. The direction of the global coordinate system axes are the same as the View Triad (shown above), but not its origin.

- Work Coordinate System (WCS)



There is one movable Work Coordinate System.

- Datum coordinate system



You can create as many datum coordinate systems as you need.

Datum coordinate systems are not addressed in this lesson. They are addressed in depth in the *Datum Features* lesson.

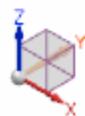
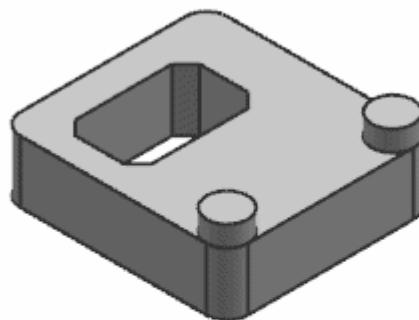
Absolute coordinate system

The *Absolute coordinate system* is a conceptual location and orientation in model space. Think of the Absolute coordinate system as X = 0, Y = 0, Z = 0. It is invisible and cannot be moved.

The Absolute coordinate system:

- Defines a fixed point and orientation in model space.
- Relates the location and orientation between different objects. For example, an object positioned at the absolute coordinates of X = 1.0, Y = 1.0, and Z = 1.0 in one particular part file, is in exactly the same absolute position in any other part file.

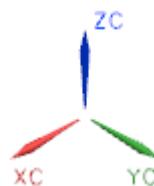
The View Triad is a visual indicator that represents the orientation of the Absolute coordinate system of the model. The View Triad is displayed in the lower-left corner of the graphics window. You can rotate a model around a specific axis on the View Triad.



WCS overview

The *Work Coordinate System* (WCS) is a right hand Cartesian coordinate system, made up of the XC, YC, and ZC axes that are 90 degrees apart from each other.

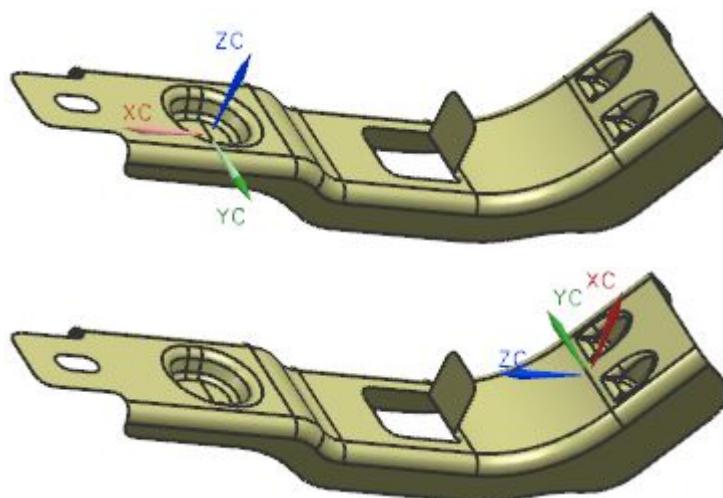
- The intersection of the axes is called the *origin* of the coordinate system.
- The origin has coordinate values of X= 0, Y= 0, Z= 0.
- The XC-YC plane of the WCS is called the *work plane*.



You can use the WCS to reference the location and orientation of objects in model space. For example you can use it to:

- Create primitives.
- Define a sketch plane.
- Create a fixed datum axis or plane.
- Create a rectangular array.

Because the WCS is a mobile coordinate system, you can move it anywhere in the graphics window to construct geometry in different orientations and locations.





- Although you can save multiple coordinate systems in a part file, only one of them can be the WCS.
- Most modeling operations in NX do not require manual manipulation of the WCS because features are added to a model relative to existing geometry.

Where do I find it?

Toolbar	All WCS commands are available on the Utility toolbar.
Menu	Format→WCS

WCS options

You can access WCS options from the **Utility** toolbar or by choosing **Format→WCS** on the menu bar.

Options available to manipulate the WCS include:

	Dynamics	Use handles to adjust the origin and orientation.
	Origin	Specify the location without changing the orientation.
	Rotate	Specify rotations in a dialog box
	Orient	Use a dialog box with Dynamic, Absolute, Current View, and several other methods.
	Set WCS to Absolute	Use to move the WCS to the position and orientation of the aht absolute CSYS.
	Change YC Direction	Use a dialog box with several options to specify the YC axis.
	Display	Show or hide the WCS.
	Save	Create a CSYS geometry entity at the current WCS origin and orientation.

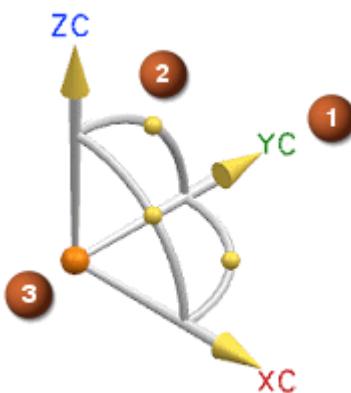


WCS Dynamics overview

Use **WCS Dynamics** to manipulate the location and orientation of the WCS.

You can enter **WCS Dynamics** at any time and it supports the **Undo** function.

1. Translation
(arrow heads)
2. Rotation
(small spheres)
3. Origin
(large sphere)



Where do I find it?

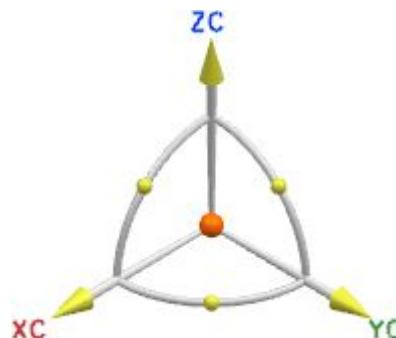
Toolbar	Utility → WCS Dynamics
Menu	Format → WCS → Dynamics
Shortcut menu	Double-click the WCS in the graphics window

Move the WCS

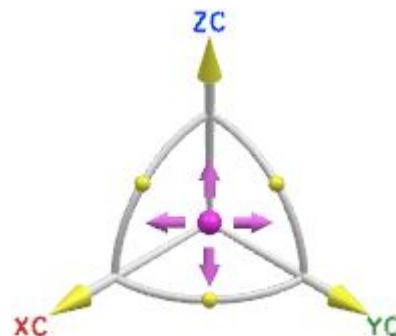
Use **WCS Dynamics**  to move the Work Coordinate System (WCS) dynamically.

1. Turn on **WCS Dynamics** in one of these ways:

- Double-click the WCS.
- On the **Utility** toolbar, click **WCS Dynamics** .
- Choose **Format→WCS→Dynamics**.



2. Select the Origin handle on the WCS and drag it to the desired location.



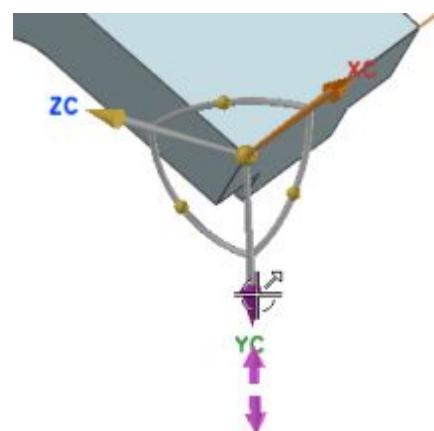
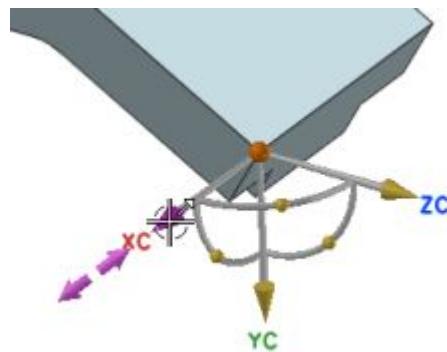
3. Turn off **WCS Dynamics** in one of these ways:

- Press Esc.
- Click the middle mouse button.
- On the **Utility** toolbar, click **WCS Dynamics** .
- Choose **Format→WCS→Dynamics**.

Activities: Coordinate systems

In the *Coordinate Systems* section, do the activity:

- *Move the working coordinate system (WCS)*



Summary: Coordinate systems

The absolute coordinate system is a stationary coordinate system that defines a fixed point in model space while the work coordinate system (WCS) is a mobile coordinate system that may be moved and reoriented as necessary to support other commands.

In this lesson you:

- Identified the difference between the absolute coordinate system and the work coordinate system.
- Relocated, rotated, and reoriented the WCS.
- Obtained geometry information relative to the WCS.

Lesson

12 Part Navigator

12

Purpose

This lesson introduces the **Part Navigator**.

Objectives

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

- Use the **Part Navigator** as a tool to understand the structure of a model.
- Reorder the features using the **Part Navigator**.

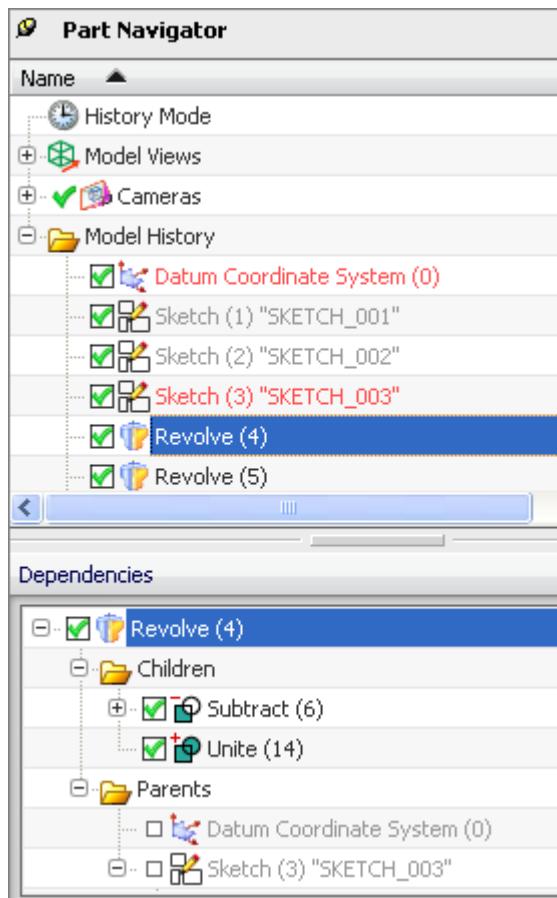


Part Navigator overview

The **Part Navigator** displays various aspects of your part in a detailed, graphical tree.

You can use the **Part Navigator** to:

- Update and understand the part's basic structure.
- Select and edit the parameters of items in the tree.
- Arrange how the part is organized.
- Display features, model views, drawings, user expressions, reference sets, and unused items in the tree.



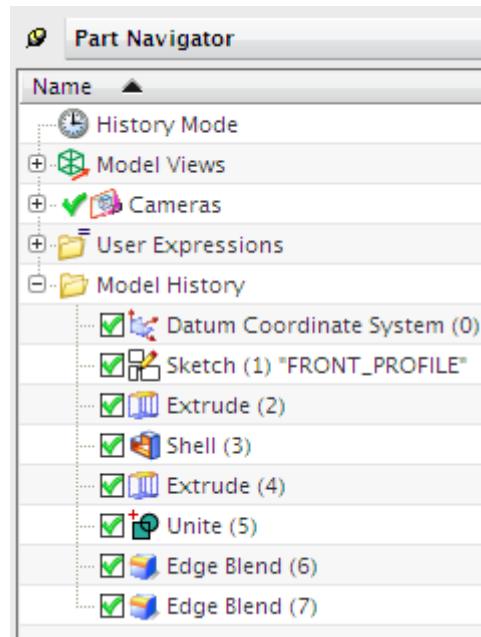
Where do I find it?

Resource bar	Part Navigator tab
Menu	Tools® Part Navigator

Main panel

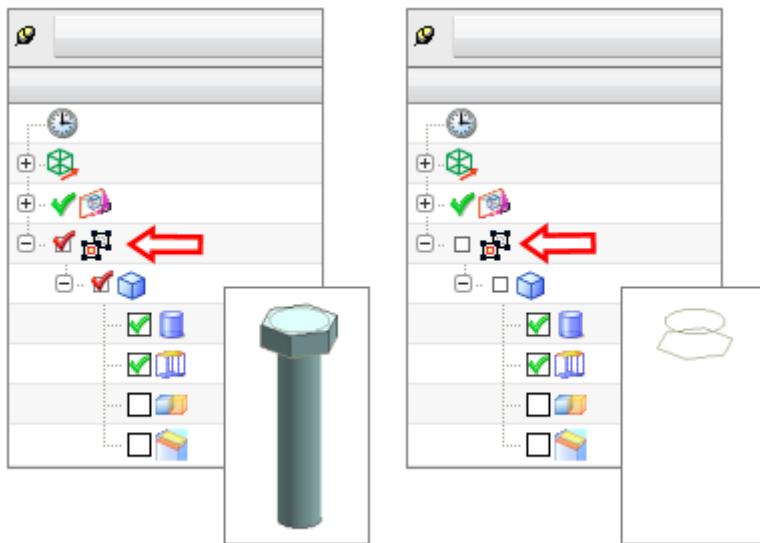
Use the main panel to see an overall graphical representation of your part's structure, to edit the parameters of items, or to rearrange the feature history.

- Double-click nodes to edit the corresponding feature.
- Select features by their nodes during dialog box interactions.
- Right-click nodes for shortcut options.
- Select or clear green check boxes to control the suppression status of features.
- Select or clear red check boxes to control the visibility of bodies.



Check boxes in the Part Navigator**☒ Red check box**

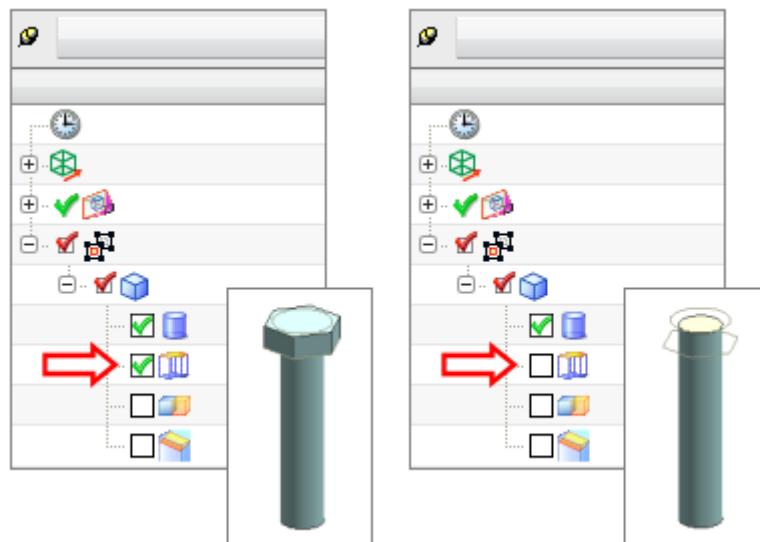
- Indicates the current **Show/Hide** status.
 - Select the red check box of an item to show it.
 - Clear the red check box to hide an item and its children.



- Not available when **Timestamp Order** is active.

Green check box

- Only features have a green check box.
- The green check box enables, or disables suppression.
 - Select the green check box to unsuppress the feature.
 - Clear the green check box to suppress the feature.



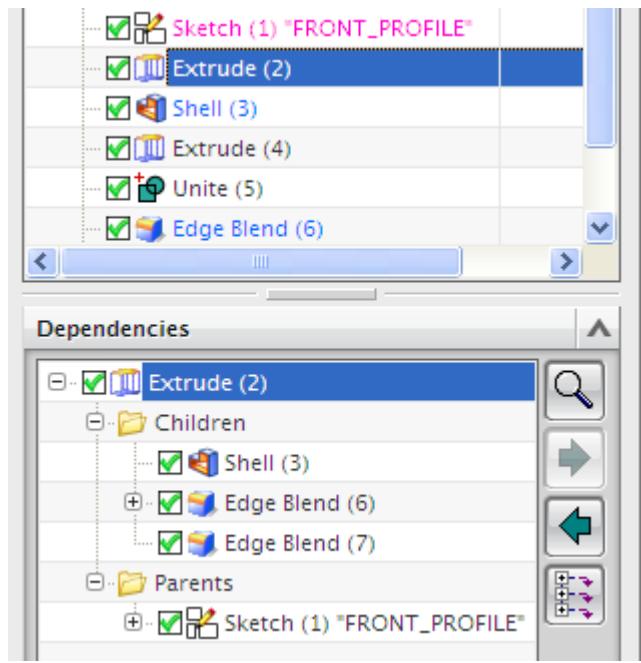
Color coding and parent/child relationships in Part Navigator

The different text colors for objects displayed in the **Part Navigator** indicate both their parent/child relationships and their **Show/Hide** status.

Color	Object relationship
Red	Parent
Blue	Child
Gray	Hidden

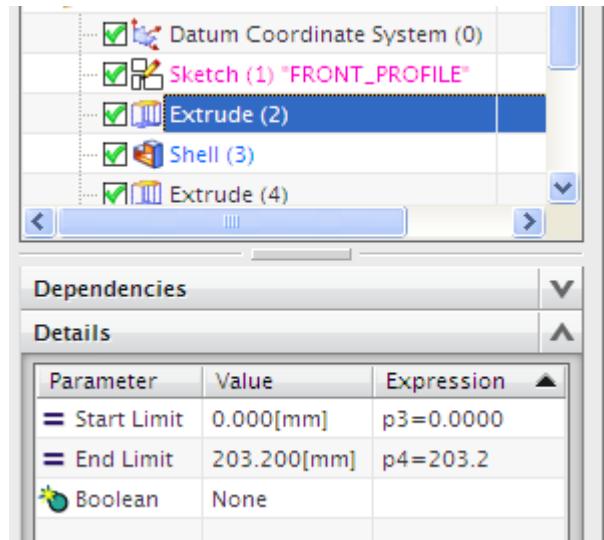
Dependencies panel

Use the **Dependencies** panel to view the parent-child relationships of the feature geometry selected in the main panel.



Details panel

Use the **Details** panel to view, and in some cases edit, the parameters belonging to the feature selected in the main panel.

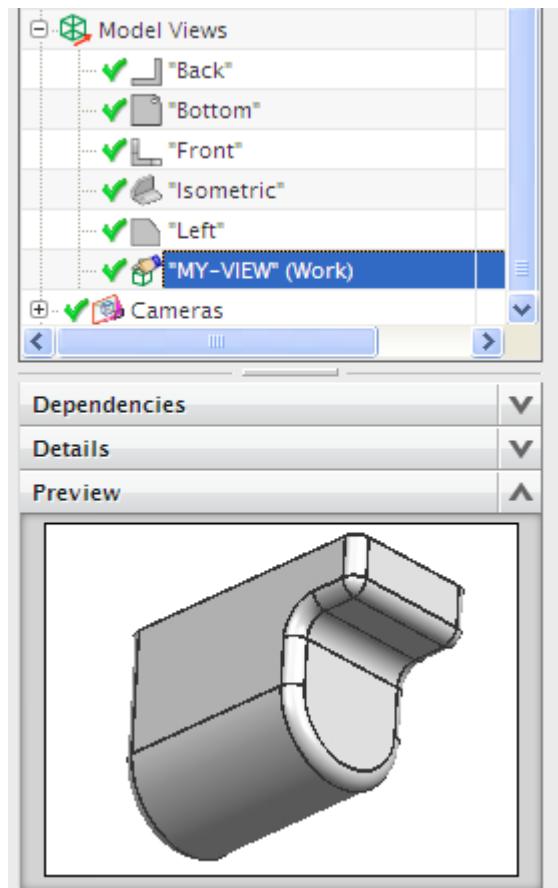


Preview panel

Use the **Preview** panel to see preview images of selected items in the main panel.



The selected item must be one that has an available preview object, such as a saved model view or drawing view.



Timestamp order

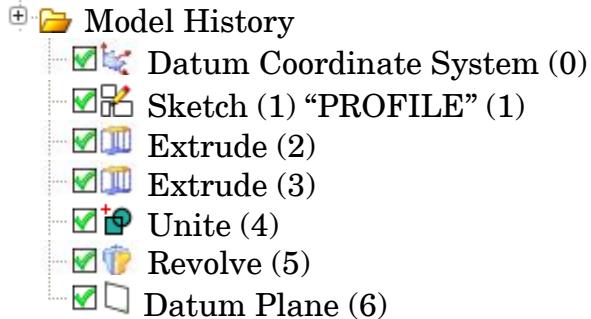
You can view the features in your work part in a timestamp order or by dependencies. The timestamp order is turned on by default.

When the timestamp order is turned on:

- All features in the work part appear in a history list of nodes in the order of their creation timestamp.
- You cannot expand or collapse feature nodes.
- You cannot view all node types.

Part Navigator

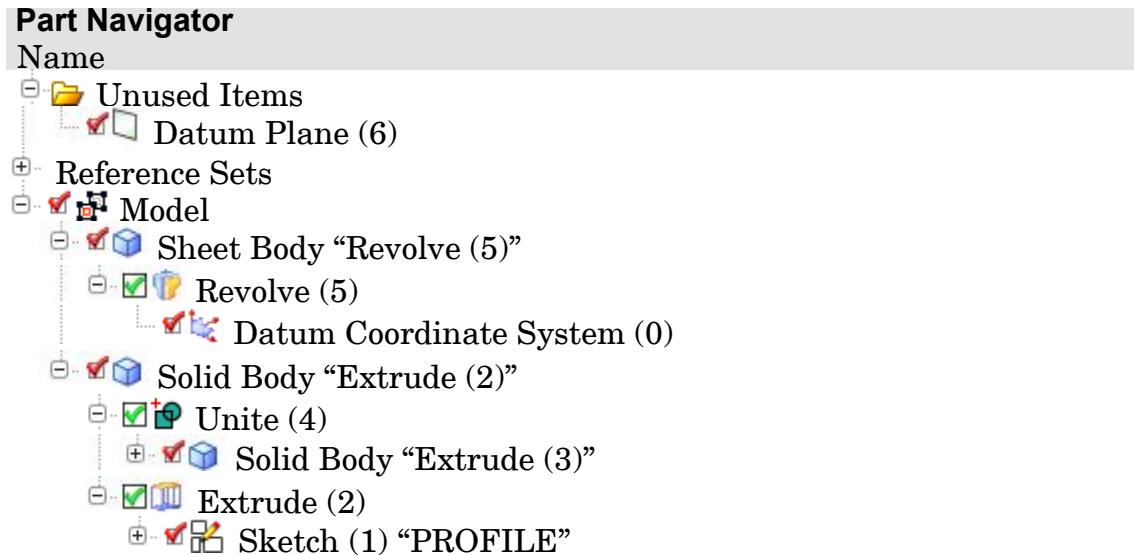
Name



The timestamp order view is not available in the History-free mode.

When the timestamp order is turned off:

- All bodies in the work part, along with their features and operations, are shown in the main panel.
- You can expand or collapse feature nodes.

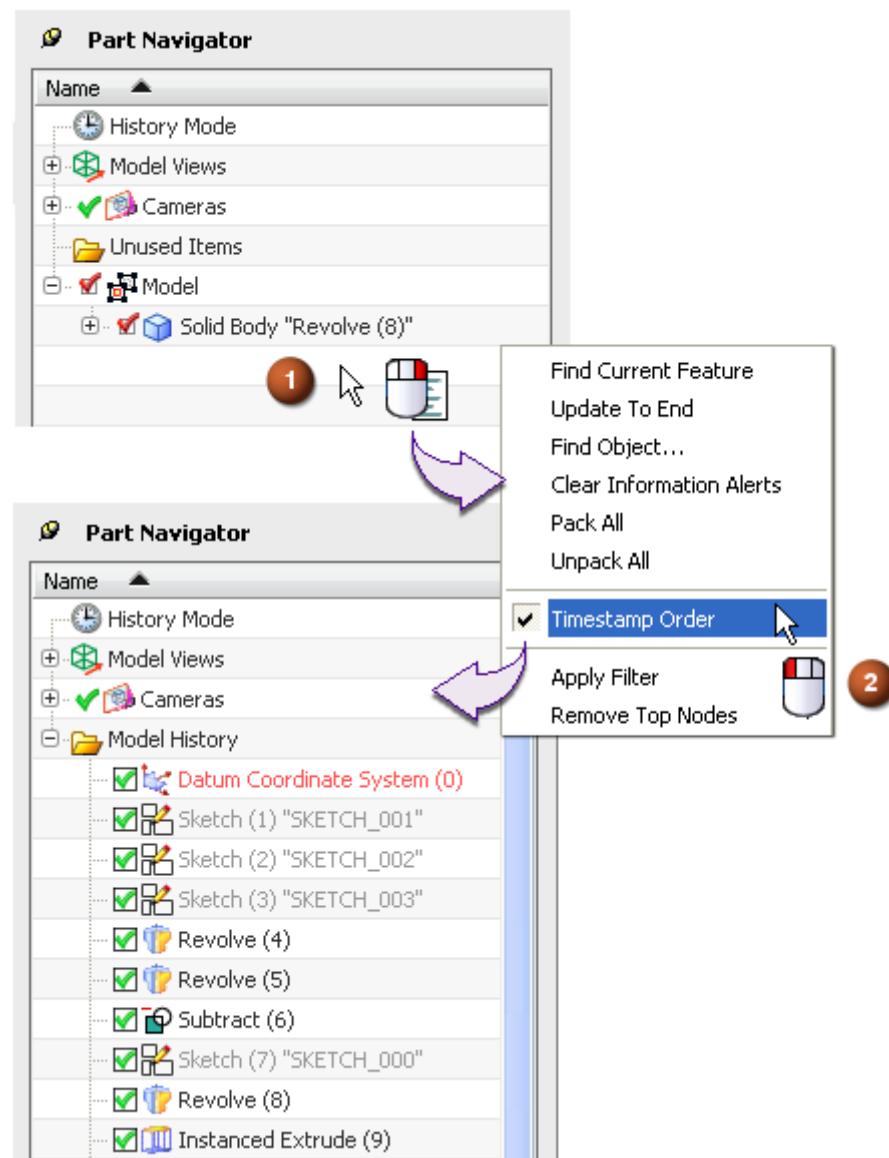


Switch between timestamp order view and design view

You can switch between the timestamp order view and the design view of the main panel of the **Part Navigator**.

To enable the timestamp order view, do one of the following:

- Choose **Tools® Part Navigator® Timestamp Order**.
- Right-click in the background of the **Part Navigator** or in the title bar, and select the **Timestamp Order** check box.



To enable the design view, do one of the following:

- Choose **Tools® Part Navigator** and turn off **Timestamp Order**.
- Right-click in the background of the **Part Navigator** or in the title bar and clear the **Timestamp Order** check box.

Part Navigator shortcut menu

Right-click a feature node in the **Part Navigator** to display a feature specific shortcut menu.



The options available depend on the type of feature you select.

Many options require the Modeling application to be active.

- **Display Dimensions** — Display the feature's parameter values until you refresh the display.
- **Show Parents** and **Hide Parents** — Display or hide parent curves, sketches, or datums.
- **Make Current Feature** — Insert new features immediately after the current feature.
- **Select Whole Branch** — Select the feature and all nodes with earlier timestamps.
- **Filter** — Simplify the display tree by hiding features by type or timestamp order.



To turn off a filter, place the cursor in the **Part Navigator** away from a feature node, right-click and choose **Apply Filter** from the shortcut menu to make it *inactive*.

- **Edit Parameters** — Edit the feature's parameters, the same as **Edit→Feature→Edit Parameters**.
- **Edit with Rollback** — Roll the model back to its state just before the feature was created, and then open the feature's creation dialog box.



Edit with Rollback is shown in bold type in the shortcut menu.
In any shortcut menu, the option in bold type is the default double-click action.

- **Suppress** and **Unsuppress** — Temporarily remove and restore a feature display from the part history.
A suppressed feature still affects *some* editing operations.
- **Reorder Before** and **Reorder After** — Change the timestamp of features.
Creation order is important to permit use of a feature as a parent, and in the **Replace Feature** command.



You can also drag nodes to valid locations.

- **Group** — Group features into a special collection called a **Feature Set**.
- **Replace** — Replace a feature's definition by another feature.
- **Make Sketch Internal** and **Make Sketch External** — Internalize or externalize a sketch that is a parent of the selected feature.
- **Edit Sketch** — Edit the parent sketch of the selected feature. This option appears only when the feature has a parent sketch.
- **Copy** — Place a copy of a feature on a clipboard.
- **Delete** — Delete the selected feature, the same as **Edit→Delete**.
- **Hide Body** and **Show** — Hide or show the body containing the selected feature.
- **Rename** — Append a user-defined name to the feature.
- **Object Dependency Browser** — Explore the parent and child relationships of features.
- **Information** — Display information about the selected feature in the Information window.
- **Properties** — Open the properties dialog box for the selected feature.
General properties include the feature name.
Attributes you assign appear in a column of the **Part Navigator**. See the online Help for details.

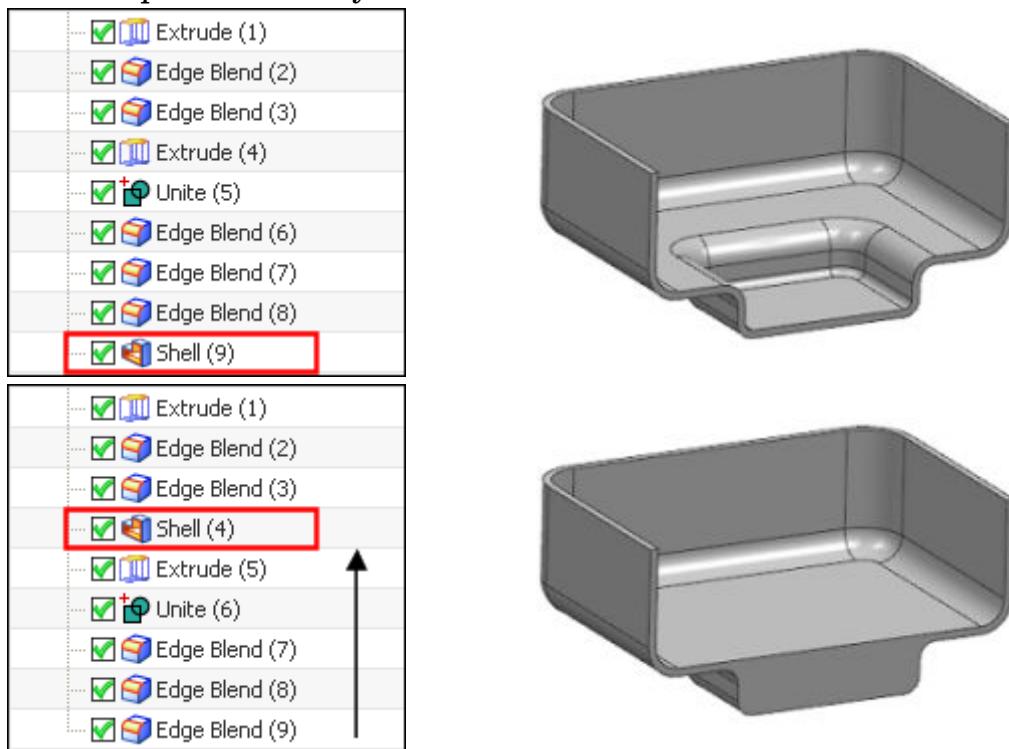


Reorder feature overview

Use the **Reorder Feature** command to change the order in which a feature is applied to a body. The desired feature can be reordered before or after a selected reference feature.

As you create features, a time stamp is assigned to each one. When you modify a body, the update follows the order of the feature time stamps.

In this example, the interior topology of a part is modified by moving the shell feature up in the history tree.



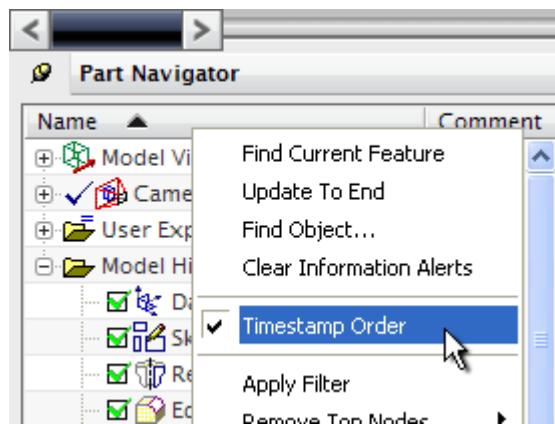
Where do I find it?

Application	Modeling
Toolbar	Edit Feature→Reorder Feature
Menu	Edit→Feature→Reorder
Shortcut menu	Right-click a feature node in the Part Navigator® Reorder .

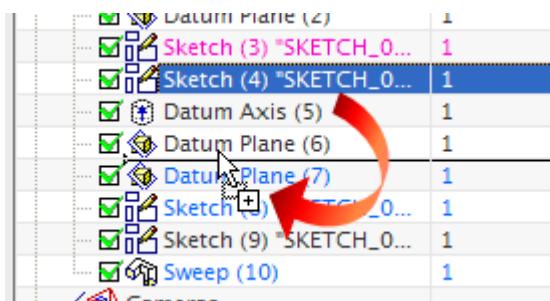
Reorder an object in the Part Navigator

To change an objects timestamp order, use this procedure to reorder an object in the Part Navigator

1. In the **Part Navigator**, right-click the column heading and select **Timestamp Order**.



2. Drag the object to reorder it to the timestamp you require.

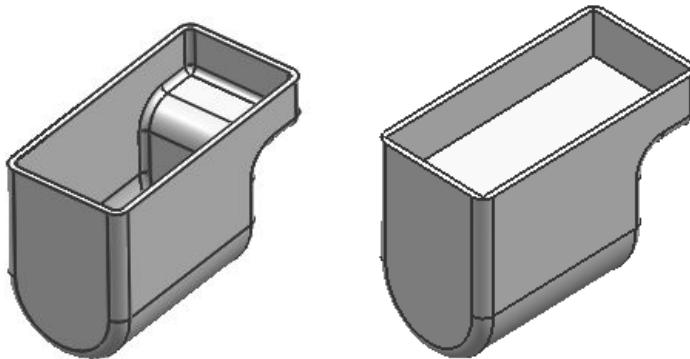


If you attempt to create a circular relationship or position an object at an invalid timestamp, your edit will be rejected.

Activities: Part Navigator

In the *Part Navigator* section, do the activity:

- *Reorder Features*



Summary: Part Navigator

You can use the **Part Navigator** to help understand the model structure and to make changes to the model.

In this lesson you:

- Accessed the **Part Navigator**.
- Reordered features.

Lesson

13 Associative copies

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Purpose

This lesson introduces commands to reuse existing geometry.

Objectives

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

- Create a rectangular and circular arrays of features.
- Mirror features and bodies.
- Copy and paste features.
- Create instances of geometry along a path.



Instance Feature overview

Use the **Instance Feature** command to:

- Create patterns such as bolt hole circles.
- Create a number of similar features such as ribs.
- Edit all **instanced features** in one step.

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You can define three Instance Feature types:

- [Rectangular Array](#)
- [Circular Array](#)
- Pattern Face

You can add edge blends, chamfers, and threads to an Instance feature.

If you create:

- An edge blend you can select **Blend All Instances**.
- A chamfer you can select **Chamfer All Instances**.
- A thread you can select **Include Instances**

When you select these options, always add the edge blend, chamfer or thread to the master feature, and not to one of the instanced features. This way, if the **array** parameters are changed, the edge blend, chamfer or thread will always remain visible in the instance set.

You cannot instance the following objects:

- Shells
- Chamfers
- Blends
- Offset sheets
- Datums
- Trimmed sheet bodies
- Instance sets
- Draft features
- Surfaces
- Trimmed features

Where do I find it?

Application	Modeling
Toolbar	Feature® Instance Feature 
Menu	Insert® Associative Copy® Instance Feature

Instance array methods

General	Create an instance array with full validation of all geometry.
	<ul style="list-style-type: none">• Instanced geometry can cross an edge of the face.• Instances can cross over from one face to another.
Simple	Create an instance array faster by eliminating excessive data validation and optimizing operations.
Identical	Create an instance array by the fastest method. <ul style="list-style-type: none">• This method does the least amount of validation.• Each instance is an exact copy of the original.• Use this method when you have many instances, and you are <i>sure</i> they are all <i>exactly the same</i>.



When you use **Simple** and **Identical**, you should make sure that all new geometry lies on the *same* face as the original feature.

If the new geometry touches or crosses the edges on the target body or any other instance, use **Analysis**→**Examine Geometry** to validate the geometry.

1. In the **Examine Geometry** dialog box, click **Set All**.
2. Select the geometry.
3. Click **Examine Geometry**.

If the array geometry fails a geometry check, click **Undo** and try a **General** array.



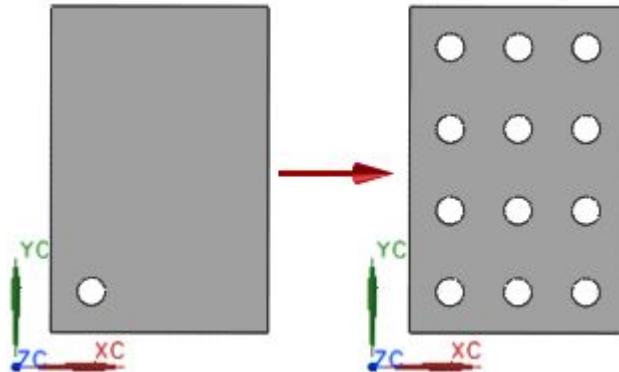
Rectangular Array overview

Use the **Rectangular Array** command to create a linear pattern of one or more selected features.

Rectangular arrays can be:

- One-dimensional in XC or YC, with one row of features.
- Two-dimensional in XC and YC, with several rows of features.

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Rectangular arrays are parallel to the XC and YC axes based on the number and offset distance you enter.

Where do I find it?

Application	Modeling
Toolbar	Feature® Instance Feature
Menu	Insert® Associative Copy® Instance Feature
Location in dialog box	Rectangular Array

Rectangular array parameters

Number Along XC	Total number of instances parallel to the XC axis, including the original feature.
XC Offset	Spacing for the instances along the XC axis.
Number Along YC	Total number of instances parallel to the YC axis, including the original feature.
YC Offset	Spacing for the instances along the YC axis.
 The number of instances for both the XC and YC directions must be a whole number greater than zero.	The offset values can be either positive or negative.

Create a rectangular array

1. In the **Instance** dialog box, click **Rectangular Array**.
2. Select the features you want to instance.
3. In the **Enter Parameters** dialog box, specify the method: **General**, **Simple**, or **Identical**.
4. Type the **Number Along XC**, **XC Offset**, **Number Along YC**, and the **YC Offset**.
5. Click **OK** to display a preview
6. Click **Yes** to create the instance array, or **No** to return to the **Enter Parameters** dialog box.

Rectangular array example

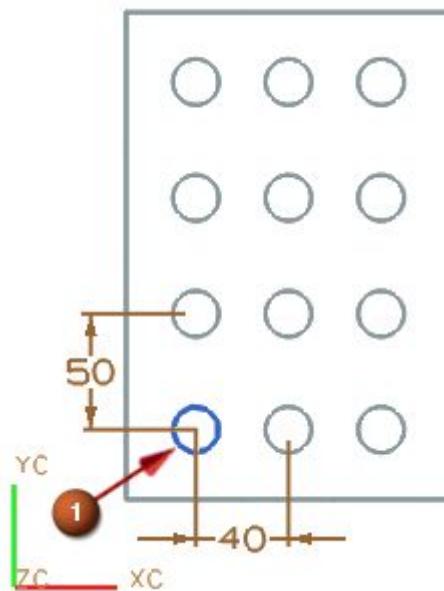
(1) Hole selected for instance

Number Along XC = 3

XC Offset = 40

Number Along YC = 4

YC Offset = 50



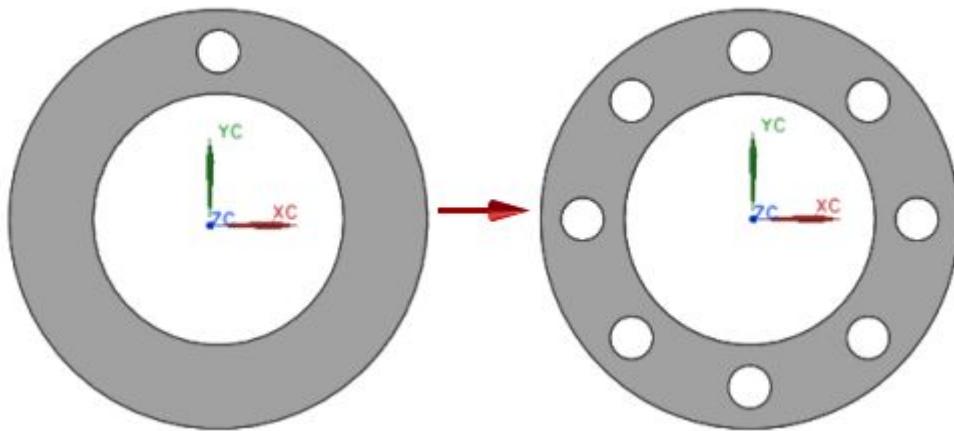


Circular Array overview

Use the **Circular Array** command to create a circular pattern of one or more selected features.

Circular arrays require the following:

- An axis of revolution.
- A reference point to revolve about.



Where do I find it?

Application	Modeling
Toolbar	Feature® Instance Feature
Menu	Insert® Associative Copy® Instance Feature
Location in dialog box	Circular Array

Circular array parameters

After you select the desired features to instance, the following options appear:

Number Total number of instances created in the circular array, including the existing feature you are instancing.

Angle The angle between the instances.



The number of instances must be a whole number greater than zero.

The angle can be either positive or negative.

Create a circular array

1. In the **Instance** dialog box, click **Circular Array**.
2. Select the features you want to instance.
3. In the **Enter Parameters** dialog box, specify the array method: **General**, **Simple**, or **Identical**.
4. In the **Number** box, type the total number of instances in the array.
5. In the **Angle** box, type the angle between instances.
6. Click **OK**.
7. Choose **Point & Direction** or **Datum Axis** to establish the rotation axis.
 - **Point & Direction** — Use the **Vector** dialog box to specify a direction and the **Point** dialog box to specify a reference point. The selected features are rotated about the reference point in a plane normal to the vector direction.

 When you use **Point & Direction**, the circular array is *not* associated to geometry you select.
 - **Datum Axis** — Select an existing datum axis.

 The circular array *is* associated to the datum axis.

The radius of the array is the distance from the rotation axis to the feature origin of the first feature you select. This radius value appears in the **Edit** dialog box.

A highlighted representation of the array is displayed.

8. Click **Yes** to create the instance array, or **No** to return to **Enter Parameters**.

Circular array example

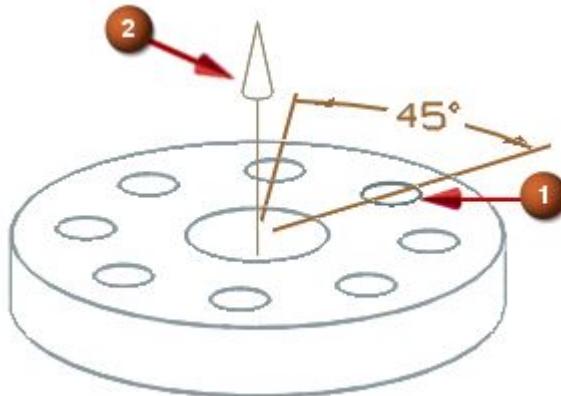
(1) Hole selected for instance.

(2) Rotation Axis (Datum Axis)

Number = 8

Angle = 45

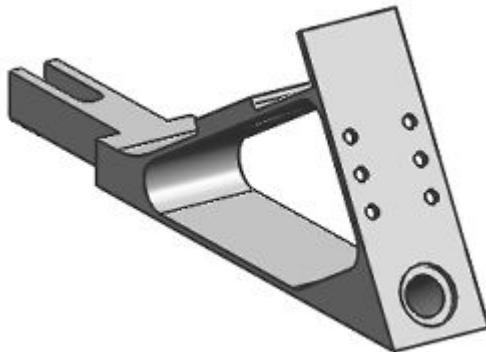
13



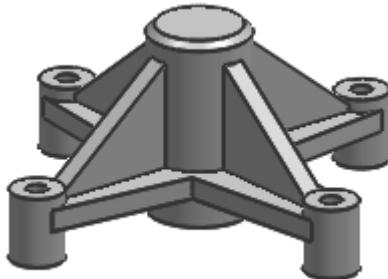
Activities: Associative copies — instance arrays

In the *Associative copies* section, do the activities:

- *Create a rectangular instance array*



- *Create a circular instance array (optional)*

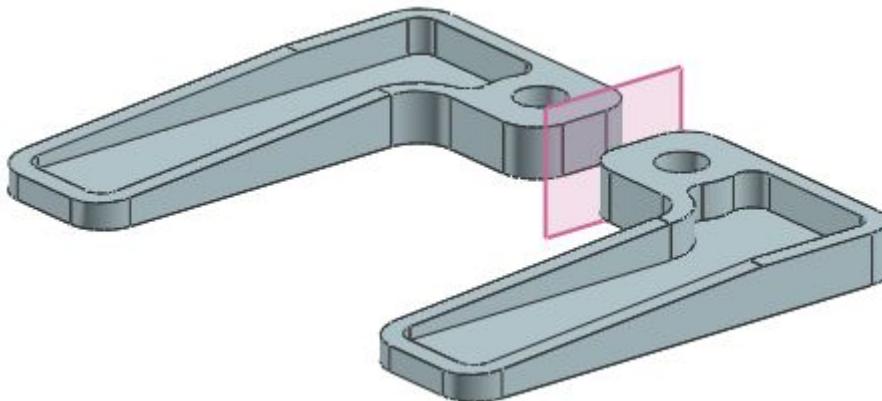




Mirror Body overview

Use the **Mirror Body** command to mirror an entire body across a datum plane. For example, you can use this command to form the other hand of a left hand or right hand part.

- When you mirror a body, the mirror feature is associative to the original body. You cannot edit any parameters in the mirrored body.
- You can specify a timestamp for the mirror feature so that any features you later add to the original body will not be reflected in the mirrored body.



Where do I find it?

Application	Modeling
Toolbar	Feature® Associative Copy Drop-down® Mirror Body
Menu	Insert® Associative Copy® Mirror Body

Create a mirrored body

1. From the menu bar, choose **Insert**→**Associative Copy**→**Mirror Body**.
2. In the **Mirror Body** dialog box, click **Select Body**  and select a body to mirror.
3. Click **Select Plane**  and select a datum plane.
4. (Optional) Clear the **Fix at Current Timestamp** check box if you want the mirrored body to reflect subsequent features added to the parent body.
5. Click **OK** or **Apply** to create the mirrored body.

Edit a mirrored body

1. Right-click the mirrored body in the graphics window or in the **Part Navigator**.
2. Choose **Edit with Rollback** from the shortcut menu
3. In the **Mirror Body** dialog box, edit the parent body, timestamp setting, or the mirror plane.

Mirror Body options

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Parent Part

Available only during edit.

Work Part — Select a parent body from the work part.

Other Part — Select a parent body from another part.
The mirrored body then becomes a WAVE linked body.

Select Body



Lets you select a body in a part to mirror.

Reverse Direction



Available only during edit.

Available only if you are mirroring a sheet body.

Reverses the surface normal of the mirrored body.

Select Plane



Select a datum plane through which to mirror a body.

Replacement Assistant

Available only during edit and only if you select geometry to replace the existing geometry.

Lets you select geometry to replace existing geometry.

WAVE Information

This group is available only during edit and only when the mirrored body is a WAVE linked body.

Parent Part displays the name of the parent part.

Object displays the name of the parent object.

Status displays the status of the WAVE link.

Fix at Current Timestamp

Select this option to *fix* the feature timestamp of the mirrored body.

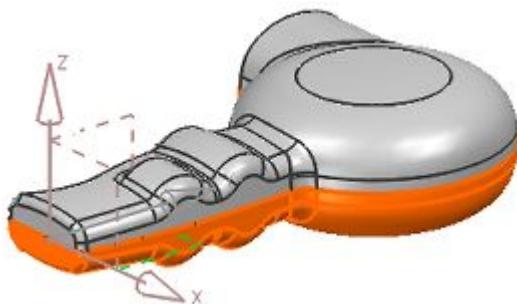
When active, only changes made to the original body *prior to the timestamp* are reflected in the mirrored body. Changes made to the original body after the timestamp are not reflected in the mirrored body.

When *not* selected, the mirrored body *dynamically changes* its location in history. Changes made to the original body are always reflected in the mirror body.

Activities: Associative copies — mirror

In the *Associative copies* section, do the activity:

- *Mirror a solid body*



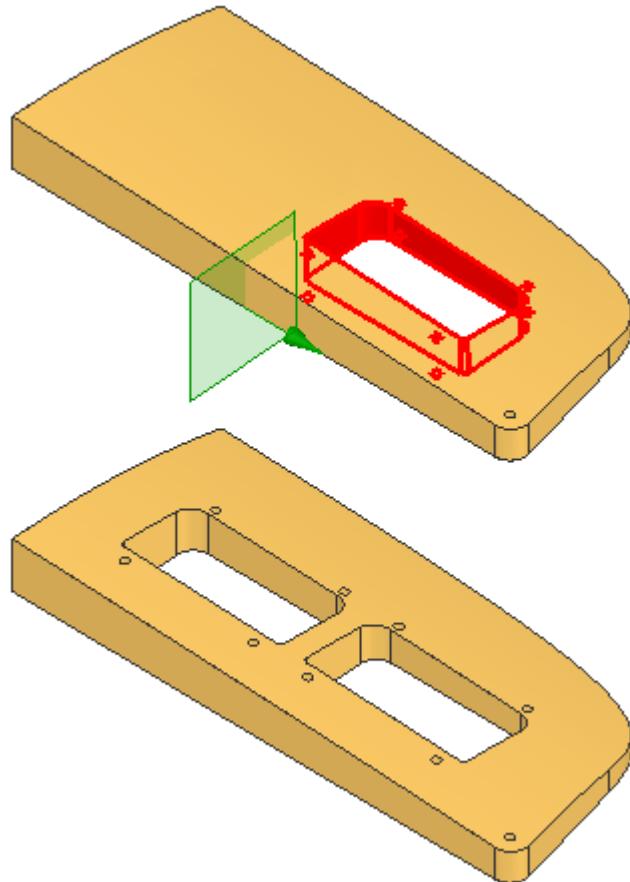


Mirror Feature overview

Use the **Mirror Feature** command to mirror one or more features within a body. Use this to build symmetrical parts.

To mirror an entire body, use the Mirror Body command.

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Extrude and hole array selected and mirrored across a datum plane

Where do I find it?

Application	Modeling
Toolbar	Feature® Associative Copy Drop-down® Mirror Feature
Menu	Insert® Associative Copy® Mirror Feature

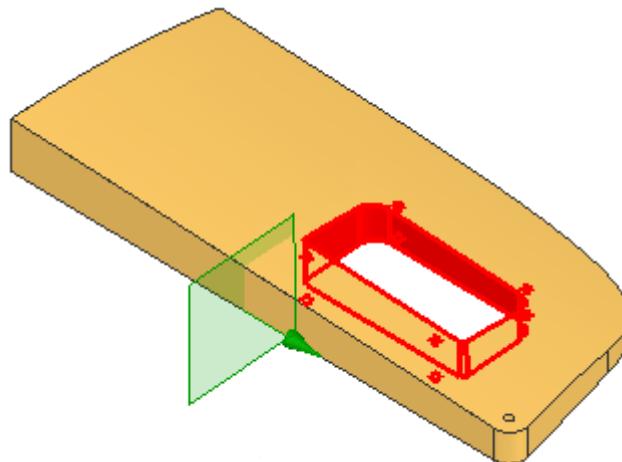


Create a mirror feature

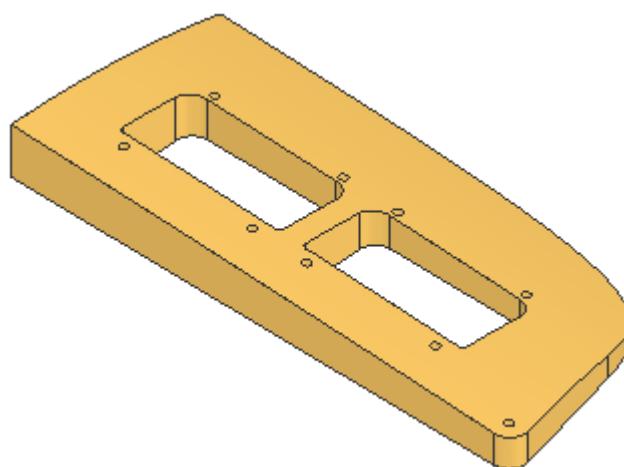
1. On the **Feature** toolbar, from the **Associative Copy Drop-down** list, select or choose **Insert® Associative Copy® Mirror Feature**.
2. Select the features to mirror.

For this example, a extrude feature and hole array are selected.

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3. In the **Mirror Feature** dialog box, in the **Mirror Plane** group, from the **Plane** option list, select an option.
For this example, **Existing Plane** is selected.
4. In the graphics window, select the existing plane.
5. Click **OK** or **Apply** to create the mirror feature.

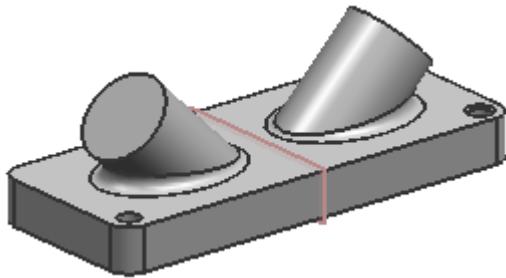


Activities: Create and edit mirror features

In the *Associative copies* section, do the activity:

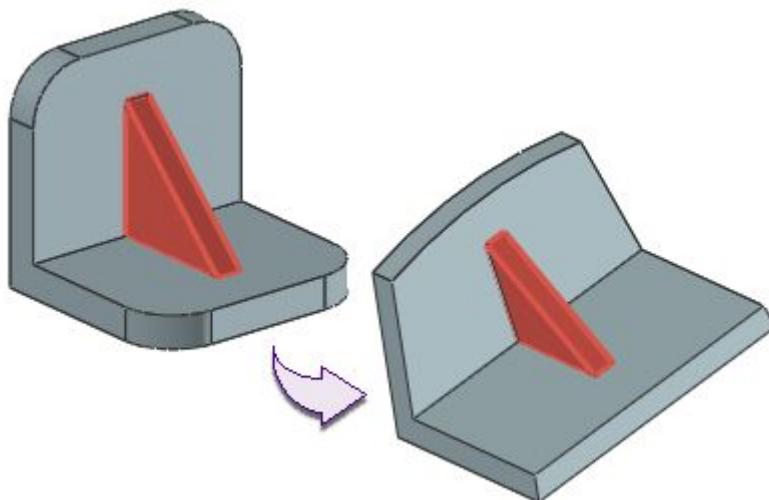
- *Create and edit mirror features*

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Copy and paste features

Use the **Copy** and **Paste** commands to copy a feature within the same part or into another part.



- You can create new expressions for the copied features or link them to the original expressions.
- You can select new parent geometry for the copied features or link them to the original parent geometry.
- When you paste a feature into another part, the original part is displayed in a separate window so that you can map the parent geometry.

You can use the **Copy Feature** command to select features from a list and copy multiple dependent features.

Where do I find it?

Application	Gateway
Toolbar	Standard® Copy / Paste
Menu	Edit® Copy / Paste / Copy Feature
Graphics window	Right-click a feature and choose Copy

Paste Feature options

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References

The list window shows all the external references for the feature you are pasting. Each unresolved reference has a (-) in front of it. As you resolve a reference (e.g., by selecting placement faces, etc., on the part where you are pasting the feature), the symbol changes to (+).



If the Parents option is Copy Original Curves, the references for the external curves are marked as resolved. (There may be other unresolved references.)

If the feature has no external references, the Paste Feature dialog box appears without the list window and Reverse Direction option.



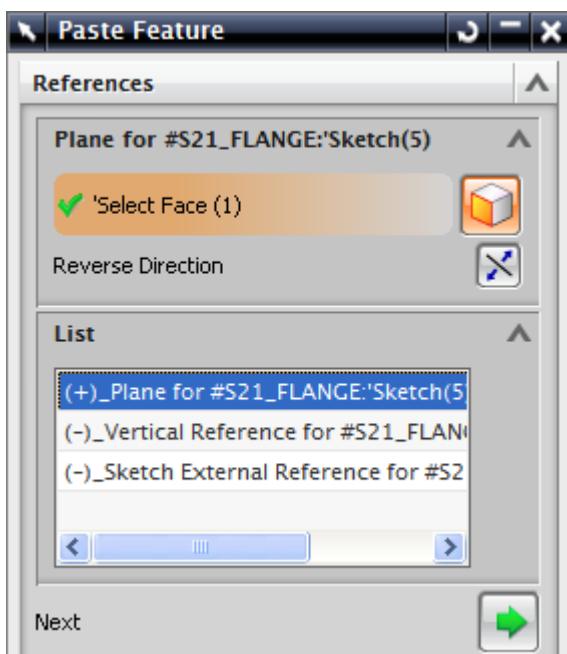
You do not always have to resolve all of the references before you can paste the feature. For example, if you copied a sweep feature with 10 curves, the unresolved references include 10 curves. But if you only supply 5 curves before choosing OK, the system may succeed in creating the sweep.

Reverse Direction

Lets you reverse the vector if you select geometry that has a direction vector.

Next

Lets you advance to the next reference in the list window.



Paste Feature settings

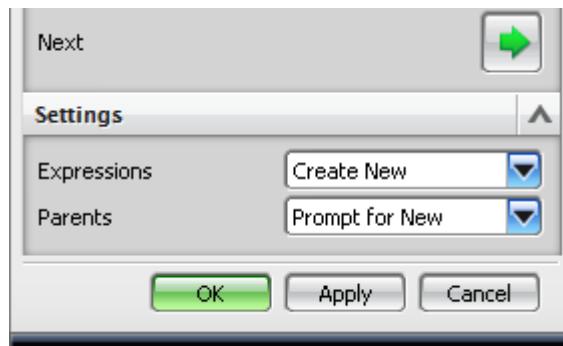
Expressions

The **Expressions** options define how the expressions in the pasted feature should be related to the original expressions:

- **Create New** creates new expressions for the pasted feature that are not associated to the expressions of the original feature.
- **Link to Original** creates new expressions for the pasted feature that are linked to the original feature (i.e. $p10=p4$).
- **Reuse Original** reuses the expressions of the original feature are used.



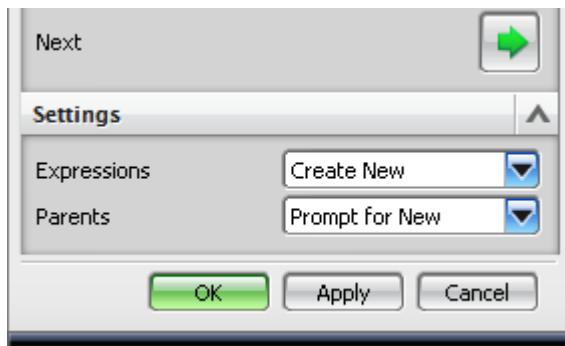
If the copy is to a different part file then Reuse Original and Link to Original both create interpart expressions.



Parents

The **Parents** options specify how parent curves of pasted features should be defined:

- **Prompt for New** the system prompts you for new curves that will replace the original curves in the pasted feature.
- **Copy Original Curves** creates a copy of the parent curves for the pasted feature.
- **Reuse Original** the parent curves of the original features are also the parent curves of the pasted feature within the same part.



Considerations when using the Copy/Paste Feature

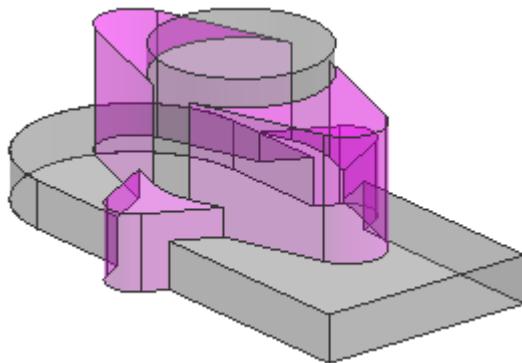
Before you use the Copy/Paste feature across part files you should evaluate the downstream impacts.

In general there are no issues to consider when the use of the Copy/Paste feature is in a single part file. When the feature is applied across part files with associativity, the result is the creation of Interpart References. This adds complexity to the design environment so caution should be used before selecting this option.

Activities: Copy and paste a sketch

In the *Associative copies* section, do the activity:

- *Copy and paste a sketch*





Instance Geometry overview

Use the **Instance Geometry** command to create associative and non-associative copies of objects.

You can create copies of:

- Bodies
- Faces
- Edges
- Curves, including sketches and points
- Datums

You can create the copies in mirror, linear, circular, and irregular patterns, as well as along a tangent continuous section.



When you edit an instance geometry feature, you can change its type, defining objects, and associative status.



Where do I find it?

Application	Modeling
	Feature Operation→Instance Geometry
Toolbar	Feature Operation→Modeling Associative Copy stack
Menu	Insert→Associative Copy→Instance Geometry

Instance Geometry types

Types are the methods you use to create instance geometry.

The available instance methods from the **Type** option list are:



From/To

Creates instance geometry by copying objects from one point or CSYS location to another point or CSYS location.



Mirror

Creates instance geometry by copying objects across a plane.



Translate

Creates instance geometry by copying objects in a specified direction.



Rotate

Creates instance geometry by rotating copies of objects around a specified point. You can add an offset distance between the rotated copies.



Along Path

(covered in this course)

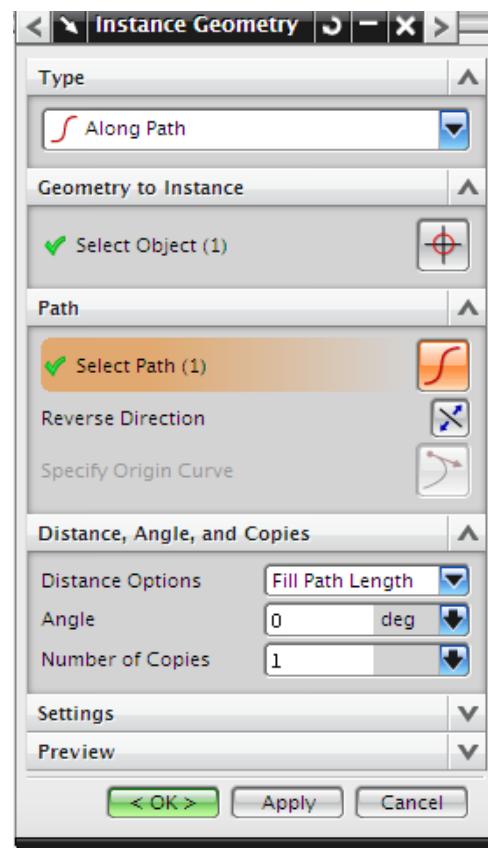
Creates instance geometry by copying objects along the path of a curve or edge. You can add an offset rotation angle to each instance.

Instance Geometry Along Path type

The following is a list of options unique to the **Instance Geometry Along Path** command.

-  **Select Path** select a tangent continuous set of curves or edges to define a path along which the copies of instanced geometry are created.
- **Distance Option Fill Path Length** the instanced geometry copies are equally distributed along the total length of the path.
- **Distance Option Arc Length** the instanced geometry copies are distributed along the path according to a parameter of arc length or percent of arc length.
- **Location** appears when **Distance Option** is set to **Arc Length**.
 - Type a value for the arc length / percent of arc length in this box, or you can drag the arc length handle located on the path to dynamically size the arc length parameter.
- **Angle** adds an incremental rotation to each copy of the instance geometry using the angle value you specify here.

- **Number of Copies** specify the number of copies (instances) of the selected geometry to create.



Activities: Geometry Instance – Along Path

In the *Associative copies* section, do the activity:

- *Instance Geometry – Along Path*

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Summary: Associative copies

You can reuse existing features and geometry to reduce the amount of time to create a model.

In this lesson you:

- Created rectangular and circular instance arrays.
- Mirrored bodies and features.
- Copied and pasted features.
- Create instances of geometry along a path

Lesson

14 Face operations

Purpose

This lesson describes various face options you may use to modify existing solid bodies and features.

Objectives

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

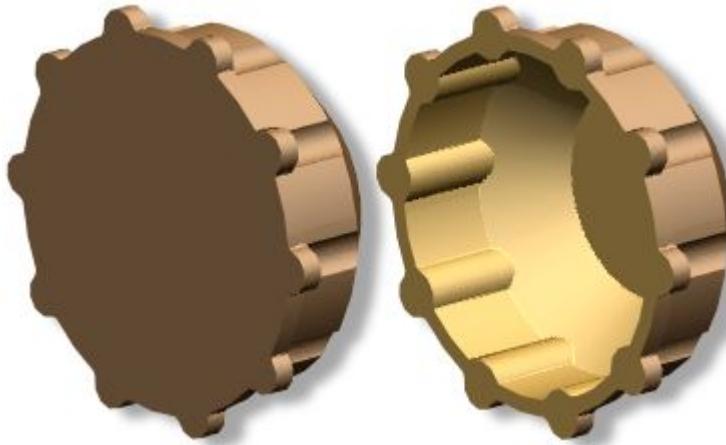
- Create a Shell feature
- Create an Offset Face feature
- Create Draft features

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Shell overview

Use the **Shell** command to hollow out a solid body or to create a shell around it by specifying wall thicknesses. You can also assign individual thicknesses to faces or remove individual faces.



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Where do I find it?

Application	Modeling
Toolbar	Feature® Shell
Menu	Insert® Offset/Scale® Shell
Shortcut menu	Right-click a solid body® Shell

Create a shell

1. On the **Feature Operation** toolbar, click **Shell** .
2. Choose the type of shell you want to create:
 - **Remove Faces, Then Shell** — Click **Select Face**  in the **Face to Pierce** group to specify one or more faces to remove from the target solid.
 - **Shell All Faces** — Click **Select Body**  in the **Body to Shell** group to select the body to shell.
3. In the **Thickness** group, type a distance value in the **Thickness** box.
4. (Optional) In the **Thickness** group, click **Reverse Direction** .
5. (Optional) Assign different thicknesses to different faces in the solid.
6. (Optional) Set or change the **Approximate Offset Faces**, **Tangent Edges**, and **Tolerance** options in the **Settings** group.
7. Click **OK** or **Apply** to create the shell.

Assign alternate thicknesses

1. Click **Select Face**  in the **Alternate Thicknesses** group and select the faces for the first face set.
2. Type a thickness value in the **Thickness n** box.

You can also drag the thickness handle or type a value in its on-screen input box.



Thickness *n* refers to Thickness 1, Thickness 2, Thickness 3, and so on.

If the direction is wrong, click **Reverse Direction**  for the face set.

3. Click **Add New Set**  to complete the current face set and begin a new set.

You can also complete the set by clicking the middle mouse button.
4. Repeat this sequence for each set of faces that require a unique wall thickness.

Shell options



You can right-click the section, preview, axis vector, or handles to quickly access many of the following options.

Option	Description
Remove Faces, Then Shell	Remove some faces of the body before shelling is done.
Shell All Faces	Shell all faces of the body.
Select Face 	Select one or more faces from a body you are going to shell. ¹ The first face selected sets the body to shell. ²
Select Body 	Select the body you want to shell. ³
Thickness	Specify a thickness for the shell walls.
(Thickness group)	Drag the thickness handle, or type a value in the on-screen input box or in the dialog box.
Reverse Direction 	Change the direction of the thickness. You can also right-click the thickness direction cone head and choose Reverse Direction , or double-click the direction cone head.
Select Face 	Select faces for a thickness set with a unique thickness value for all faces in the set. ² Complete the set by clicking Add New Set or by clicking the middle mouse button. You can add as many face sets as the model allows.
Thickness <i>n</i> (Alternate Thickness group)	Specify an independent thickness value for the currently selected thickness set in the List . You can drag the face set handle, or type a value in the on-screen input box or dialog box.
	The Thickness <i>n</i> label changes to match the currently selected Thickness set; Thickness 1, Thickness 2, etc.

1. Appears only when the Type is Remove Faces, Then Shell.

2. Selection Intent for faces is available.

3. Appears only when the Type is Shell All Faces.

Option	Description
Add New Set	Complete the current face set.
	You can also complete the current face set by clicking the middle mouse button.
List	Thickness sets appear in the list with their name, value, and expression information.
	To select a thickness set, click its on-screen input box in the graphics window or click its entry in the List .
	Delete a thickness set in the list.
	You can also delete a thickness set by right-clicking it in the list and choosing Delete or by right-clicking its handle and choosing Delete .
Approximate Offset Faces	Require NX to repair self-intersections caused by offsetting surfaces in the body, by approximating the face within the specified Tolerance .
	Use this option for complicated surfaces that would fail due to self-intersections during shell creation.
Tangent Edges	Extend Shelf Face at Tangent Edge — Allow the creation of edge faces along smooth boundary edges.
	Extend Tangent Face — Prevent the creation of edge faces along smooth boundary edges.
Tolerance	Enter a new tolerance value here to override the modeling distance tolerance for the shell operation.

Selection Intent face rules

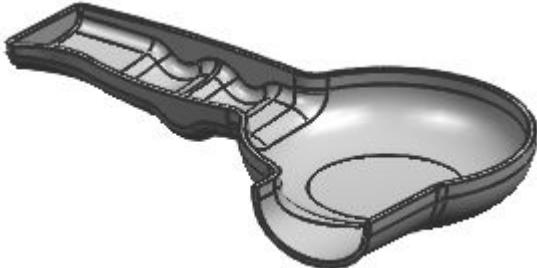
When a feature requires a collection of faces, **Face Rule** options are available. The list displays the face selection rules that are applicable to the feature you are creating.

Rule	Description
Single Face	Single-select one or more faces as a simple list of objects without intent.
Tangent Faces	Select a single face that acts as the seed of a collection of smoothly connected faces.
Adjacent Faces	Collect all faces that are immediately adjacent to the single face you select.
Face and Adjacent Faces	Collect the selected face and faces that are immediately adjacent to the single face you select.
Feature Faces	Collect all faces produced by the feature responsible for the face you select.
Region Faces	Specify a region of faces. Select a single seed face, and then specify the boundary faces.
Tangent Region Faces	Select a seed face and then, <i>optionally</i> , one or more boundary faces.
Body Faces	Collect all faces of the body containing the single face you select

Activities: Shell

In the *Face Operations* section, do the activities:

- *Create a shell*





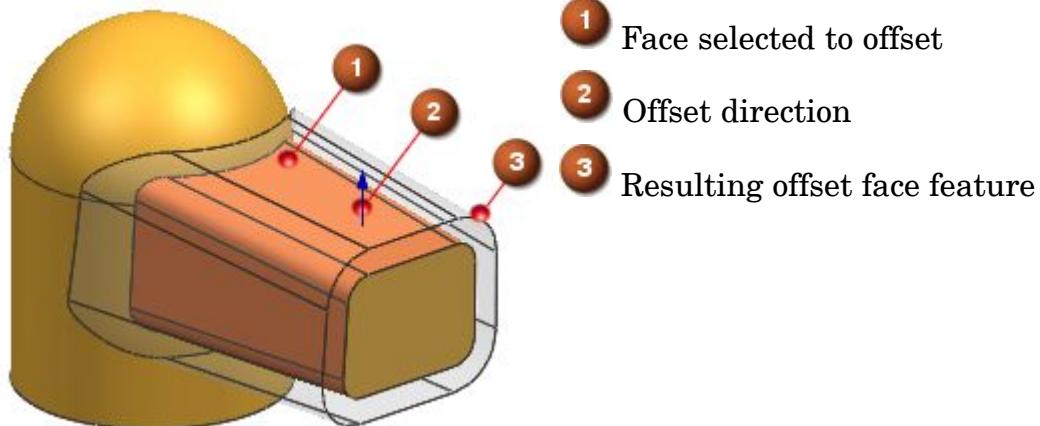
Offset Face overview

Use the **Offset Face** command to offset one or more faces along the face normals.

You can offset faces either by positive or negative distances, provided the topology of the body does not change. You can add a single **Offset Face** feature to multiple bodies.



The **Thicken** command is similar to the **Offset Face** command. You can use the Boolean options with the **Thicken** command but you can only add or remove material with the **Offset Face** command.



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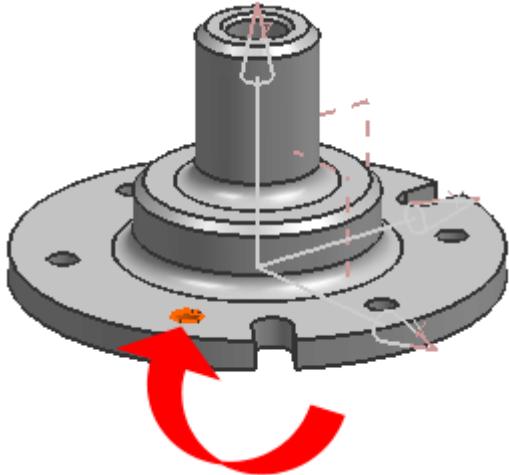
Where do I find it?

Application	Modeling
Toolbar	Feature® Offset Face
Menu	Insert® Offset/Scale® Offset Face

Activities: Offset a face

In the *Face operations* section, do the activity:

- *Offset a face*



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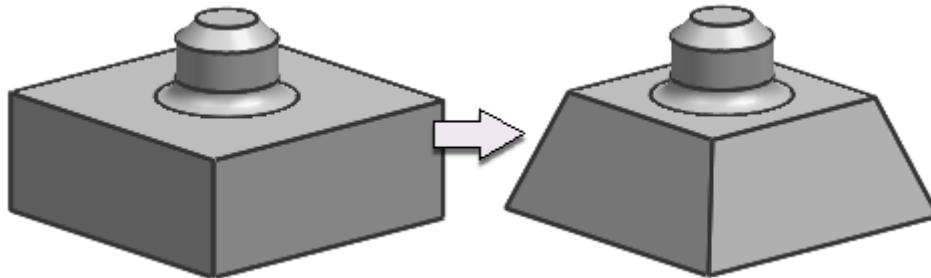


Draft overview

Use the **Draft** command to apply a draft to faces or bodies relative to a specified vector.

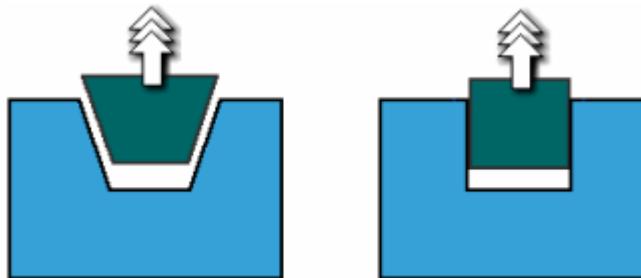
You can do the following:

- Specify multiple draft angles and assign an angle to a set of faces.
- Add a single Draft feature to multiple bodies.



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The **Draft** command is typically used to apply slope to faces for the use in molded, or die cast parts, so that when the mold or die separates, the faces move away from each other rather than sliding next to each other.



Generally, the draw direction is the direction the mold or die must move to be separated from the part. However, if you are modeling a mold or die, it is the direction the part must move to be separated from the mold or die.

Where do I find it?

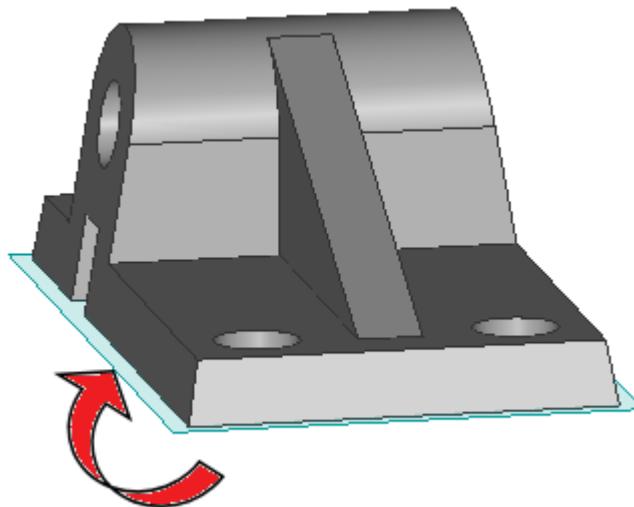
Application	Modeling, Shape Studio
Toolbar	Feature® Draft
Menu	Insert® Detail Feature® Draft

Draft types

You can create the following four types of draft using the **Draft** command.

From Plane

If the draft operation requires that a planar cross section through the part be maintained throughout the face rotation, then use the **From Plane** type. This is the default draft type selected when you open the Draft dialog box for the first time.

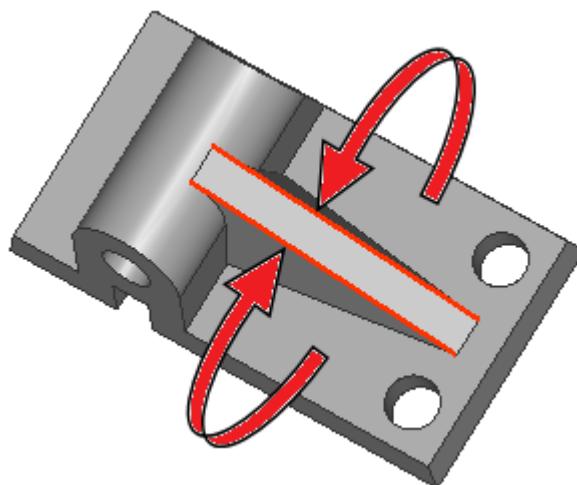


From Edges

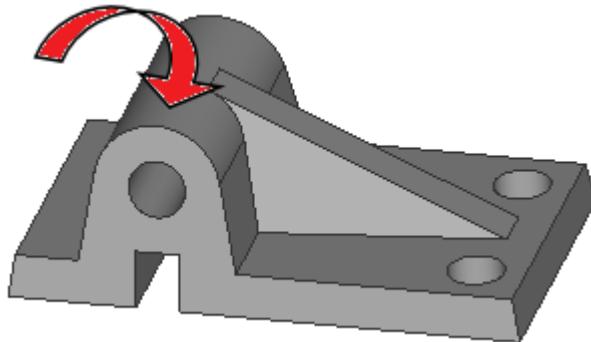
If the draft operation requires that edges be maintained throughout the face rotation, then use the **From Edges** type.



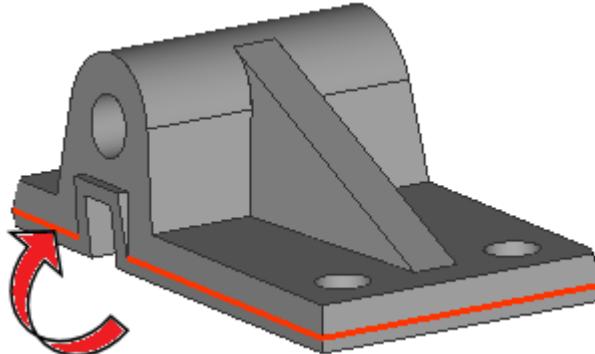
This is the only draft type that can have varying draft angles within a face using **Variable Draft Points**.



Tangent to Faces If the draft operation requires that the face selected to be drafted maintain tangency with an adjacent drafted face, then use the **Tangent to Faces** type.



To Parting Edges If the draft operation requires that a planar cross section through the part be maintained throughout the face rotation, and that a ledge be created as necessary at parting edges, then use the **To Parting Edges** type.



Draw Direction

Regardless of the draft type selected, you must always specify a draw direction.

If you are modeling a mold or die, it is the direction in which the mold or die must move to be separated from your model of the molded part.

The draft angle is positive if the normal of the face to be drafted has a component vector *along* the draw direction.

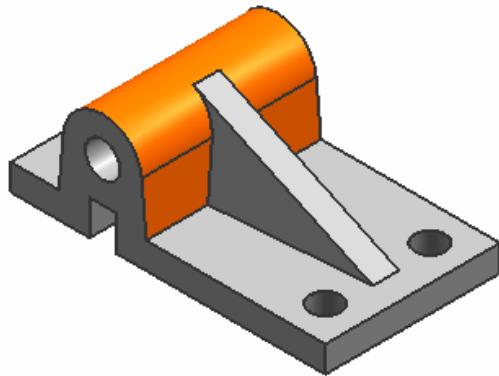
In following image positive draft is shown on the left and negative draft is shown on the right.



Activities: Draft

In the *Face operations* section, do the activities:

- *Add draft from faces and edges*



Summary: Face operations

In this lesson you:

- Created a Shell feature.
- Created Offset features.
- Created Draft features.

Lesson

15 Edge operations

Purpose

This lesson introduces the edge operation commands to provide additional definition to the edges of a model. These commands include **Edge Blend** and **Chamfer**.

Objectives

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

- Create constant and variable radius edge blends.
- Use overflow options to control blend intersections.
- Create chamfers.

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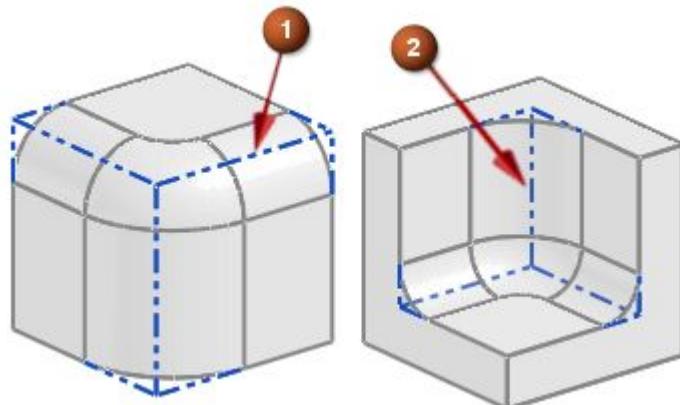


Edge Blend overview

Use the **Edge Blend** command to round sharp edges between faces.

This command operates like a ball that rolls along an edge, maintaining contact with the faces that meet the edge.

The blending ball rolls on the inside of faces to round the edges, removing material (1), and the outside of faces to fillet the edges, adding material (2).



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Where do I find it?

Application	Modeling
Toolbar	Feature® Edge Blend
Menu	Insert® Detail Feature® Edge Blend
Graphics window	Right-click edges® Blend

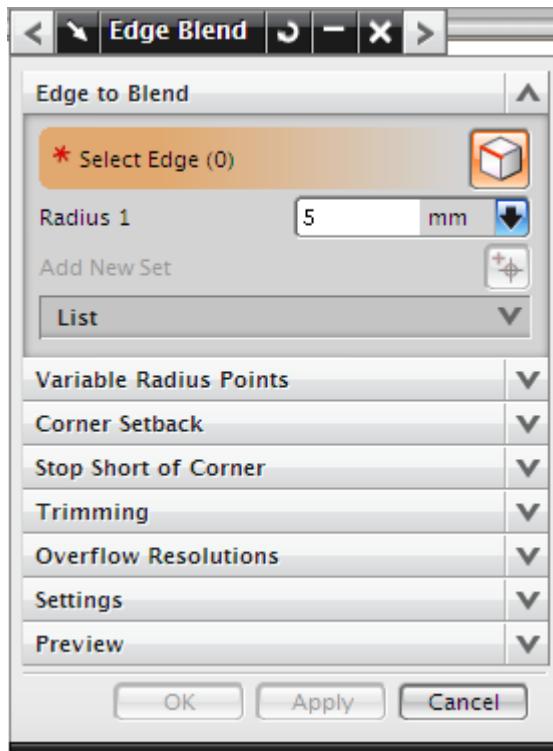
Edge Blend dialog box

After you click **Edge Blend** a dialog box is displayed and you are prompted to select a set of edges. You can type the radius in the **Radius n** box.



Radius *n* refers to **Radius 1**, **Radius 2**, **Radius 3**, and so on.

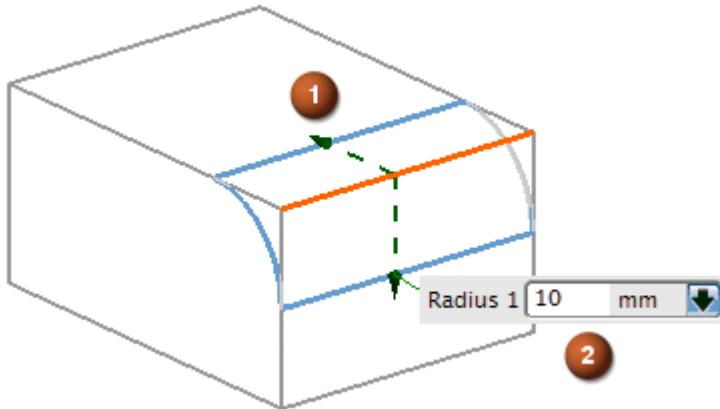
Use a **Curve Rule** to collect related edges or to speed up selection.



Edge Blend preview

As you select edges, the preview is updated. If the preview fails, it means the blend will probably also fail. You should see a warning window explaining the problem.

Adjust the radius by dragging one of the radius drag handles (1) or by typing the value in the dynamic input field (2).



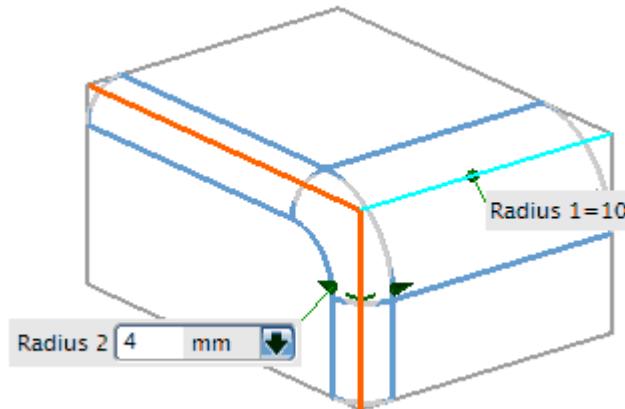


Add New Set

A single blend feature may consist of one or more sets of edges. Each set may have a different radius value.

Click **Add New Set** in the dialog box (or click the middle mouse button once) to select another set of edges.

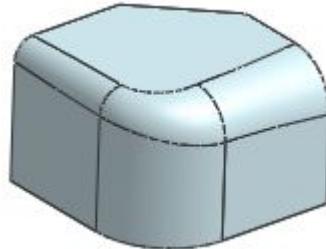
You may continue to define another edge set or complete the blend operation by clicking **OK**.



Edge Blend options

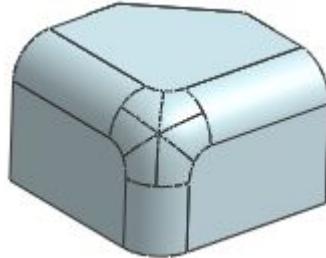
Variable Radius Points

Apply unique radius values at selected locations on an edge blend.



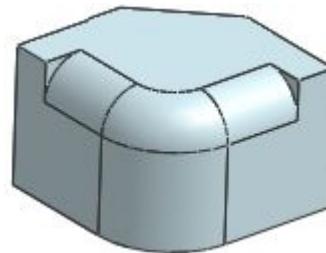
Corner Setback

Select a corner end point and increase the corner radius values along each edge.



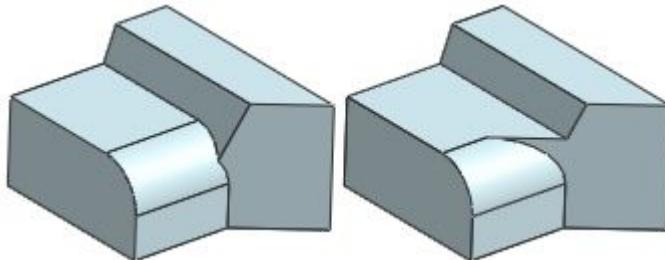
Stop Short of Corner

Stop an edge blend at a point short of the end of the edge.



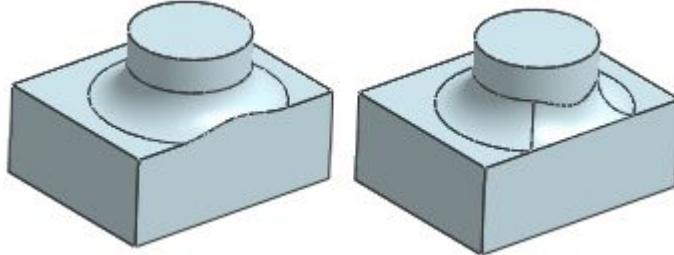
Trimming

Trim an edge blend to a selected face or plane.



Overflow Resolutions

Control how blend overflows are handled. Blend overflow occurs when tangent edges of a blend encounter other edges on the body.



Resolve blended edge overflow

Blend overflow occurs when tangent edges of a blend encounter other edges on the solid.

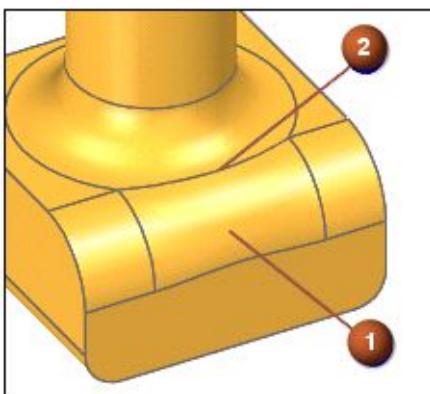


Try combinations of these options to get different results.

Resolution	Description
Roll Over Smooth Edges	The blend extends onto a smoothly connected (tangent) face that it encounters.
Roll on Edges (Smooth or Sharp)	The blend to foregoes tangency with one of the defining faces, and rolls onto any edge, whether smooth or sharp.
Maintain Blend and Move Sharp Edges	The blend to maintains tangency with the defining faces, and moves any encountered edges to the blend face.
Select Edge to Force Roll on	Select an edge on which you want to force the software to apply the Roll On Edges (Smooth or Sharp) option.
Select Edge to Prohibit Roll on	Select an edge on which you want to prevent the software from applying the Roll On Edges (Smooth or Sharp) option.

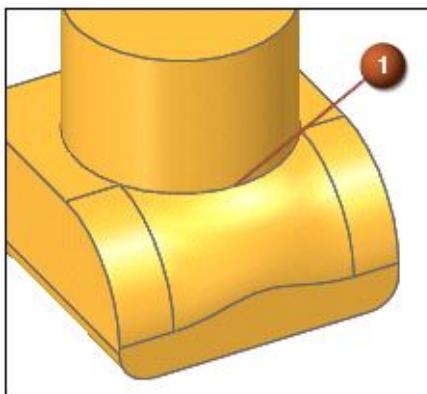
Allowed Overflow Resolution examples

Roll Over Smooth Edges



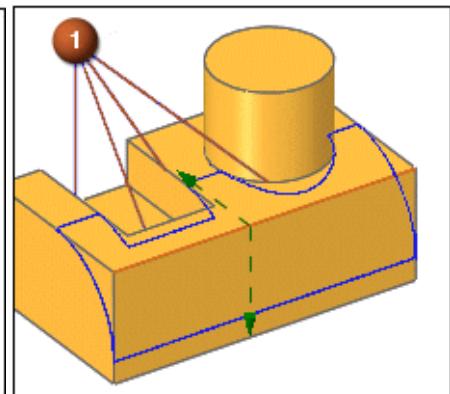
A blend that overflows the edge of an existing blend (1) produces a smooth, shared edge where the blends meet (2).

Roll on Edges (Smooth or Sharp)



A blend that encounters an existing edge, foregoes tangency and leaves the existing edge unchanged (1).

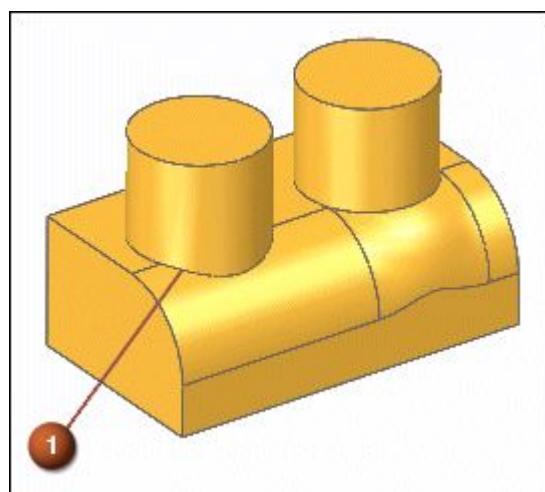
Maintain Blend Over Sharp Edges



A blend that encounters existing sharp edges, maintains tangency and moves the existing edges (1).

Explicit Overflow Resolutions

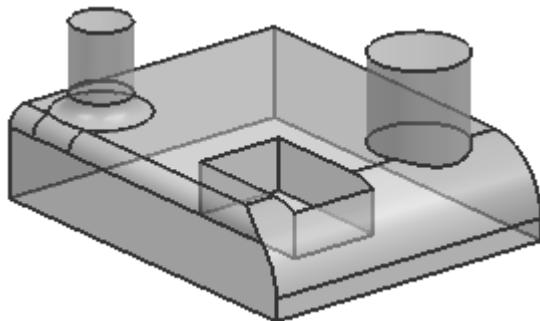
For this edge blend, an encountered edge (1) is selected with **Select Edge to Prohibit Roll on**, to not have the **Roll On Edges (Smooth or Sharp)** option applied to it. The edge of the other cylinder is *not* prohibited and *is* processed by the **Roll On Edges (Smooth or Sharp)** option.



Activities: Allowed Blend Overflow Resolutions

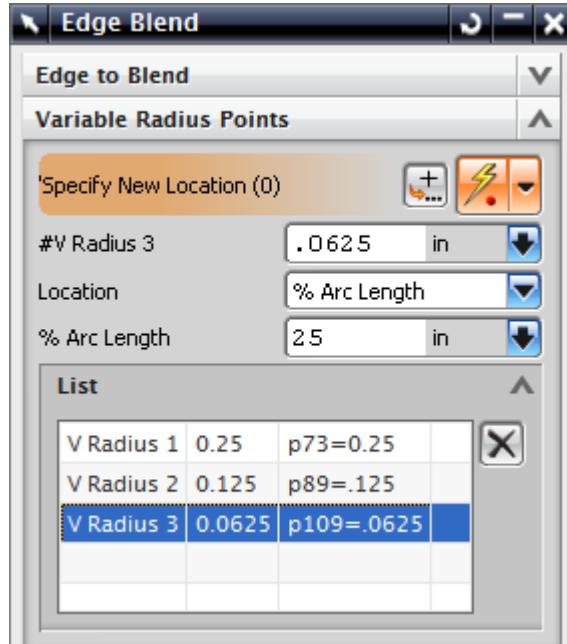
In the *Edge operations* section, do the following activity:

- *Allowed Blend Overflow Resolutions*



Variable radius blends

You can create a variable radius blend by specifying the radius at multiple points along the blend's edge set.



Creating variable radius blends

The Selection bar Snap Points can help you specify points.



You can change the position of a variable radius point by:

- Dragging the point.
- Entering the desired value in the **% Arc Length** or **Arc Length** on-screen input box.
- Entering the desired value in the **Edge Blend** dialog box.

You can switch between **% Arc Length** (the default) and **Arc Length** by:

- Right-clicking the variable point.
- Changing the **Location** option in the dialog box.

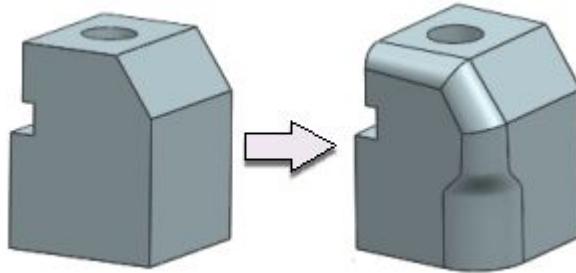
You can change the blend radius at a variable point by:

- Entering a new value in the dialog box or on-screen input box.
- Dragging a point handle.

To delete a point, right-click the point or point handle in the graphics window and choose **Remove**. The point may also be selected and deleted in the **Variable Radius Points** list from the **Edge Blend** dialog box.



Create an edge blend of variable radius

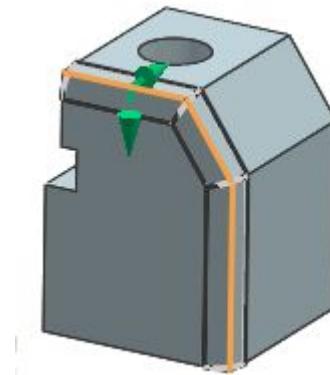


This example shows how to create a blend with varying radius by defining points along the edge at which you want to vary the radius of the blend.

1. On the **Feature** toolbar, click **Edge Blend** or choose **Insert® Detail Feature® Edge Blend**.

In the **Edge Blend** dialog box, **Select Edge** is active.

2. Select edges for the edge set.



For this example, three edges are selected for the edge set.

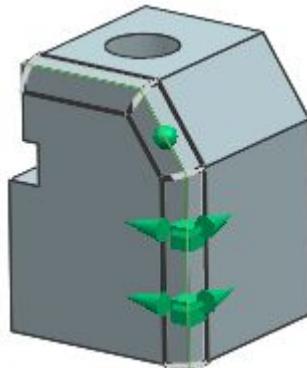
3. In the **Radius 1** box, type a value to specify the radius

For this example, it is set to 10.

4. In the **Variable Radius Points** group, click **Specify New Location**.

5. Click **Point**  and specify points on the edges where you want to set a variable radius value.

For this example, two variable radius points are selected on a vertical edge.



6. To specify variable radius values, in the **Variable Radius Points** group, do the following:

- a. Click **List**.

The list shows the two selected variable radius points, **V Radius 1** and **V Radius 2**.

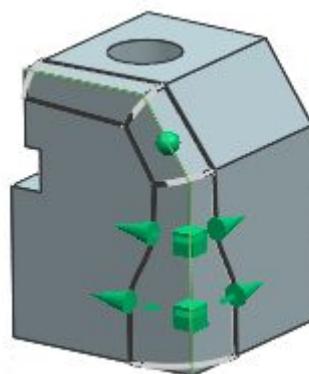
- b. Select **V Radius 1** and in the **V Radius 1** box, type a value.

For this example, it is set to 15.

- c. Select **V Radius 2** and in the **V Radius 2** box, type a value.

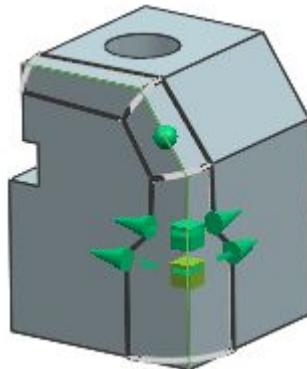
For this example, it is set to 25.

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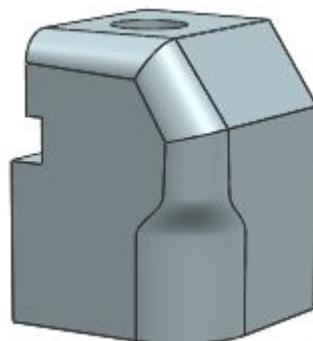
7. Move the location of a variable radius point by choosing a **Location** option: **%Arc Length**, **Arc Length**, or **Through Point**.

For this example, **%Arc Length** for **V Radius 2** is set to 45.



If you manually move the point location by changing the value for **% Arc Length**, the point loses its associativity.

8. Click **OK** or **Apply** to create the blend feature with variable radius points.

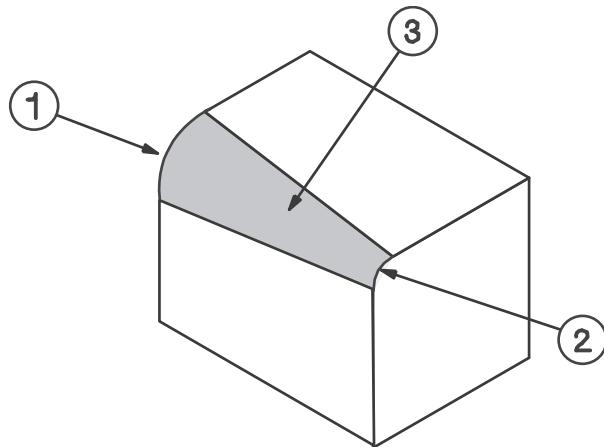


Variable blend tips and techniques

- If you do not give enough information to create the blend, the system infers information for you depending on other selected geometry.
- If you do not provide a point and radius for a selected edge, the system uses the default radius to create the blend for that edge.

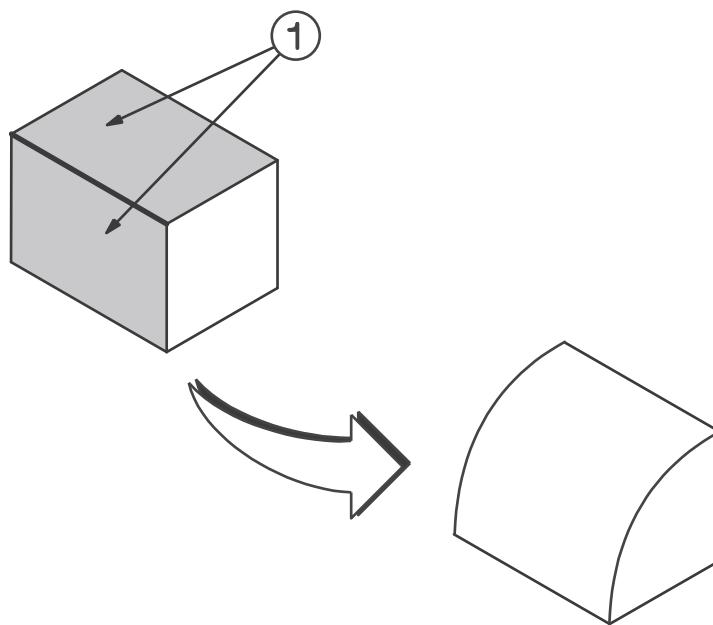
The following are some rules used to produce the desired blends:

- To produce a linearly varying blend (3), define a different radius at each end of the edge (1,2).



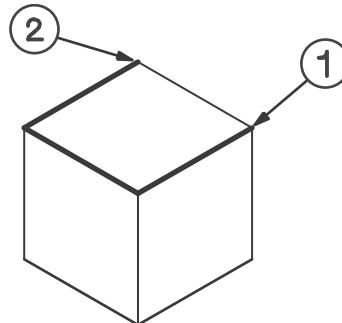
15

- If you must perform an operation that will blend away entire faces (1), blend only one edge at a time.

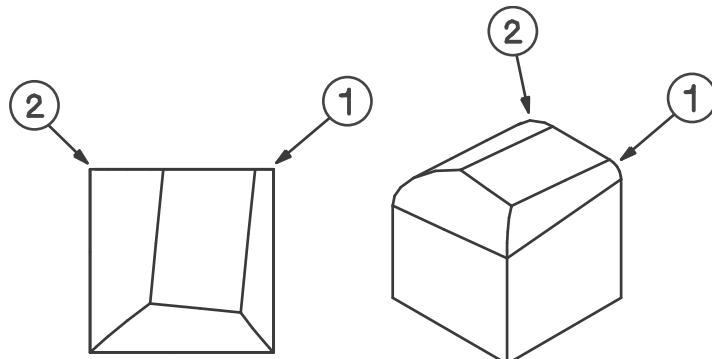


- If you select an open loop set of edges and supply radii only to the two open endpoints, the blend will vary continuously from endpoint to endpoint.

In the example below, three edges on the top face of the block are blended. A radius of 0.1 is assigned at end point (1) and a radius of 0.4 is assigned at end point (2).

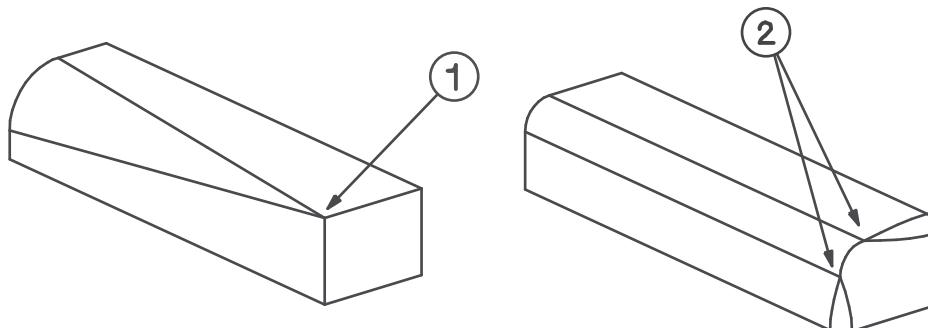


The result is shown below in both a TOP and ISO view.



15

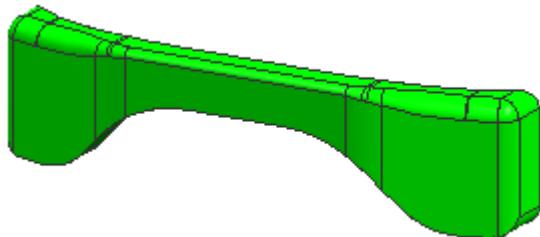
- You can create a variable radius blend with the radii value of zero at one of the selected vertices (1,2).



Activities: Variable radius blends

In the *Edge operations* section, do the activity:

- *Create a variable radius blend*

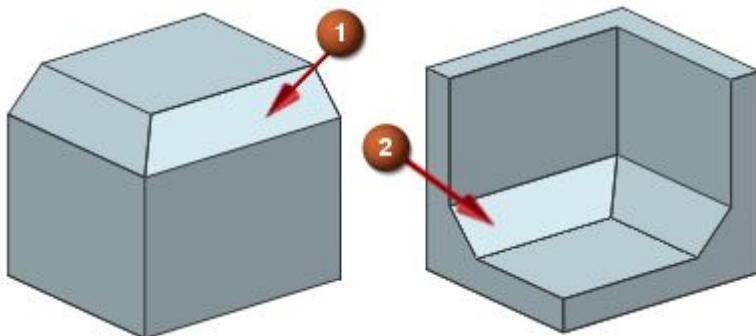




Chamfer overview

Use the **Chamfer** command to bevel the edges of one or more bodies.

Depending on the shape of the body, the chamfer bevels edges by subtracting material (1) or adding material (2).



You can define the cross section of the chamfer by specifying:

- One symmetric offset distance.
- Two offset distances.
- An offset distance and an angle.

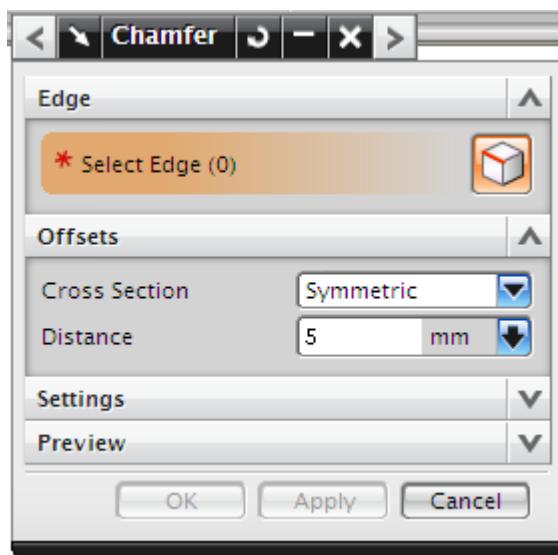
15

Where do I find it?

Application	Modeling
Toolbar	Feature® Chamfer
Menu	Insert® Detail Feature® Chamfer
Graphics window	Right-click an edge of a body® Chamfer

Create a Chamfer

1. On the **Feature Operation** toolbar, click **Chamfer** , or choose **Insert→Detail Feature→Chamfer**.
2. Select one or more edges.
3. In the **Offsets** group, specify an option from the **Cross Section** list; **Symmetric**, **Asymmetric**, or **Offset and Angle**.
4. In the dialog box, type offset values that correspond to the cross section option.
5. (Optional) In the **Settings** group, specify an option from the **Offset Method** list, **Offset Edges along Faces**, or **Offset Faces and Trim**.
6. (Optional) In the **Settings** group, select **Chamfer All Instances**, if the chamfered edge is, or may be, instanced.
7. (Optional) In the **Preview** group, select **Preview** to preview results, or clear it to show only the drag handles.
8. (Optional) Use drag handles or on-screen input boxes to modify offsets.
9. (Optional) In the **Offset** group, click **Reverse Direction** to flip the chamfer.
10. Click **OK** or click the middle mouse button to create the chamfer.



Chamfer options



You can change the **Cross Section** option or click **Reverse Direction** in the dialog box, or, you can use the shortcut menu over a drag handle.

Edge

Select Edge

Select one or more edges from the *same* body, using a Curve Rule.

Offsets

Symmetric — Create a simple chamfer, using a single, positive offset from a selected edge along both of its faces.

Cross Section

Asymmetric Create a chamfer using two positive values for the edge offsets.

Offset and Angle — Create a chamfer whose offsets are determined by one positive offset value and a positive angle.

Type a distance value for the offset when the **Cross Section** is **Offset and Angle** or **Symmetric**.

Distance

You can also drag the distance handle to specify the value.

Distance 1

Type distance values when the **Cross Section** is **Asymmetric**, or drag the handles.

Distance 2

Type an angle value for the angle when the **Cross Section** is **Offset and Angle**.

Angle

You can also drag the angle handle to specify the angle.

Move the offsets or the offset and angle from one side of the chamfer edge to the other.

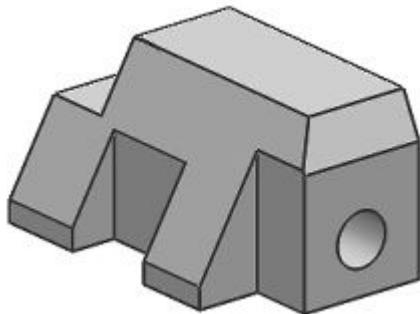
Reverse Direction

Not available when the cross section is symmetric.

Activities: Edge operations — chamfers

In the *Edge operations* section, do the activity:

- *Create chamfers*



Summary: Edge operations

Use the **Edge Blend** and **Chamfer** commands to alter the edges of a solid body. All of the blended edges or chamfered edges created in a single operation are considered to be one feature.

In this lesson you:

- Blended edges using a constant radius.
- Blended edges by assigning by varying the radius at different locations.
- Used overflow options to control blend intersections.
- Chamfered edges using different input options.

Lesson

16 Basic freeform

Purpose

Most designers need more control than they can achieve while only using *analytic* shapes. Spline Curves and sheet bodies take design beyond analytic geometry so you can freely construct any form you require, thus the term “freeform.”

Objectives

In this introductory lesson you will learn how to:

- Create splines
- Create a sketch on path
- Create a Variational Sweep feature

16



Studio Spline overview

A spline is a standard curve in most CAD systems. Unlike lines and conic curves, the spline can be adjusted to virtually any shape in two or three dimensions.

A splines flexible nature and variety of data interpretation methods make splines the foundation of freeform modeling.

Studio Splines interactively create associative or non associative splines.

Studio splines were developed for the aesthetic designer who needs to watch a curve develop as the definition progresses. You can modify Studio Splines by dragging either defining points or poles.

Spline creation methods

These are the basic methods you can use to create splines:

By Poles

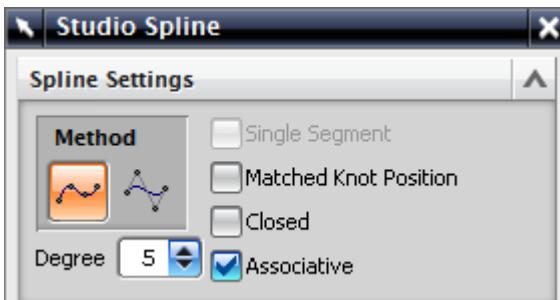
— the spline gravitates towards each data point (pole) but passes through only the two endpoints.

This method lends it self well to aesthetic designs.

Through Points

— the spline passes through a set of data points.

This method is well suited when there are a small amount of precise data points through which a curve must pass.



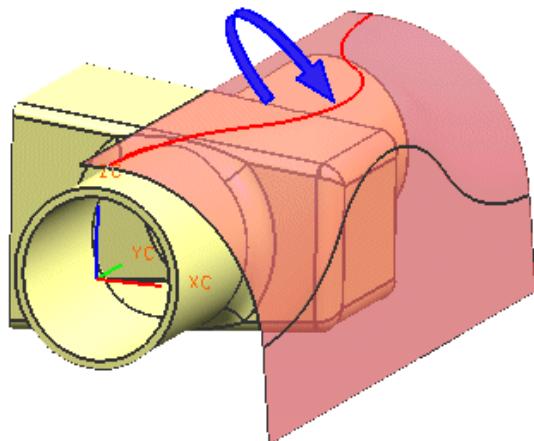
Where do I find it?

Application	Modeling
Toolbar	Curve→Spline
Menu	Insert→Curve→Studio Spline

Activities: Create a spline

In the *Basic Freeform* section, do the activity:

- *Create a spline*

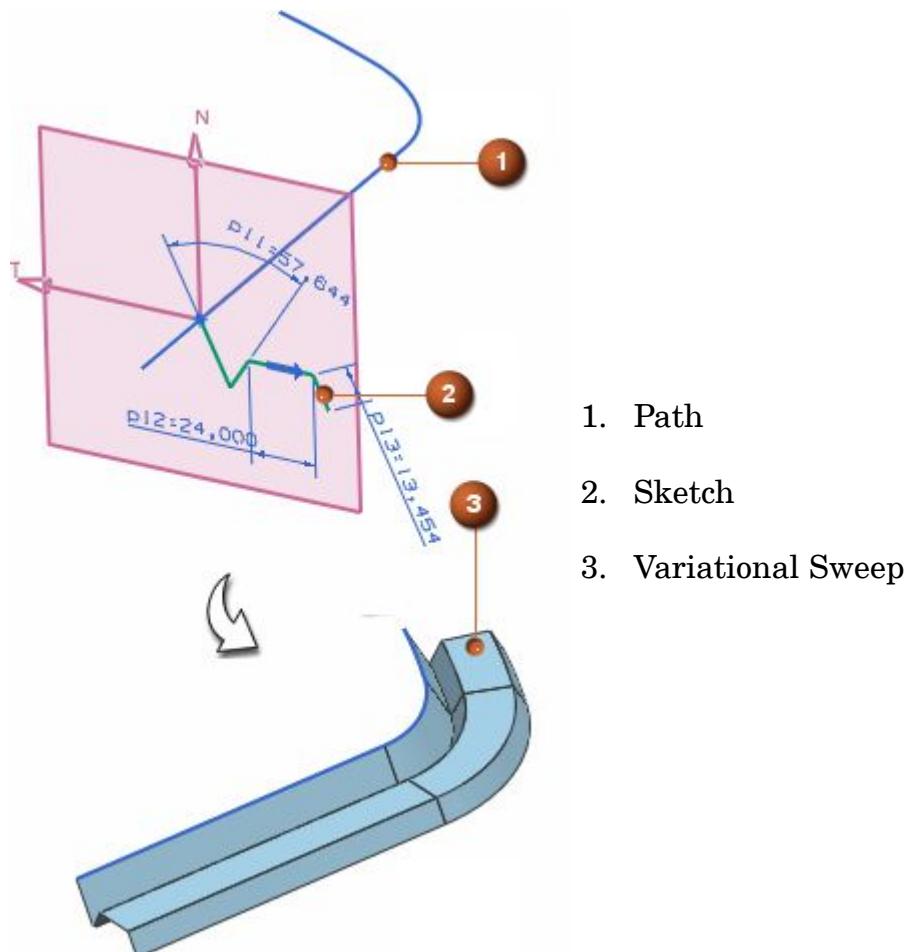




Sketch On Path

The **Sketch On Path** sketch type creates a datum plane perpendicular to a string of curves or edges and a sketch with origin and orientation related to both the path and the datum.

Create a sketch on path when you are building an input profile for features like **Variational Sweep**.



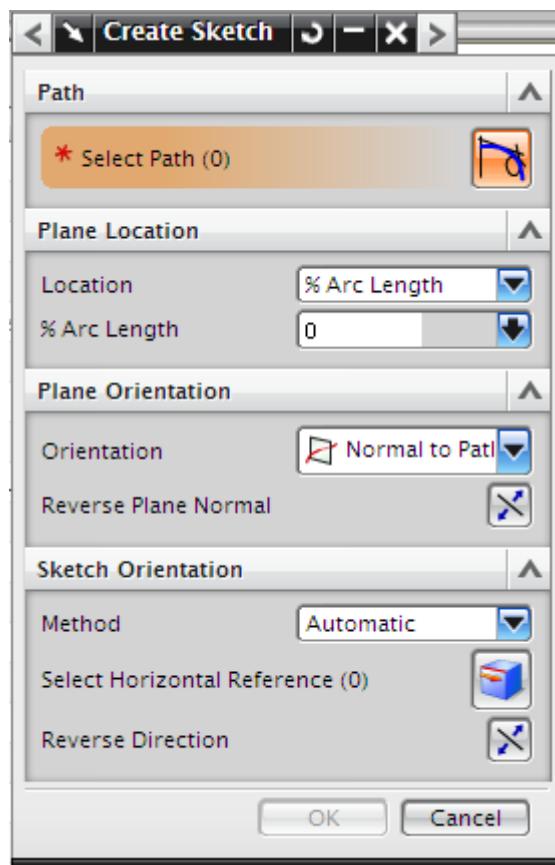
Sketch on Path dialog box

The following is an overview of the unique **Sketch on Path** creation options.

- **Path** defines a sketch on a path. This option is embedded within modeling commands such as **Variational Sweep**.
- **Plane Location** specifies how you want to define the plane location along the path.
 - **% Arc Length** starts the sweep at a specified percentage.
 - **Arc Length** starts or ends the sweep at a specified length.
 - **Through Points** starts or ends the sweep at a specified point on the guide curve.
- **Plane Orientation** specifies the direction of the sketch plane.
 - **Normal to Path** sets the sketch plane normal to the path on which you are sketching.
 - **Normal to Vector** sets the sketch plane normal to a specified vector using the Vector dialog.
 - **Parallel to Vector** sets the sketch plane parallel to a specified vector using the Vector dialog.
 - **Through Axis** aligns the sketch plane to a specified axis using the Vector dialog.

- **Sketch Orientation**

- **Relative to Face** orients the sketch, either implicitly or explicitly assigned, to a face.
- **Use Curve Parameters** the sketch orients using curve parameters.
- **Automatic** NX orients sketch axes using curve parameters. If you select an edge, NX orients the sketch axes relative to the face.



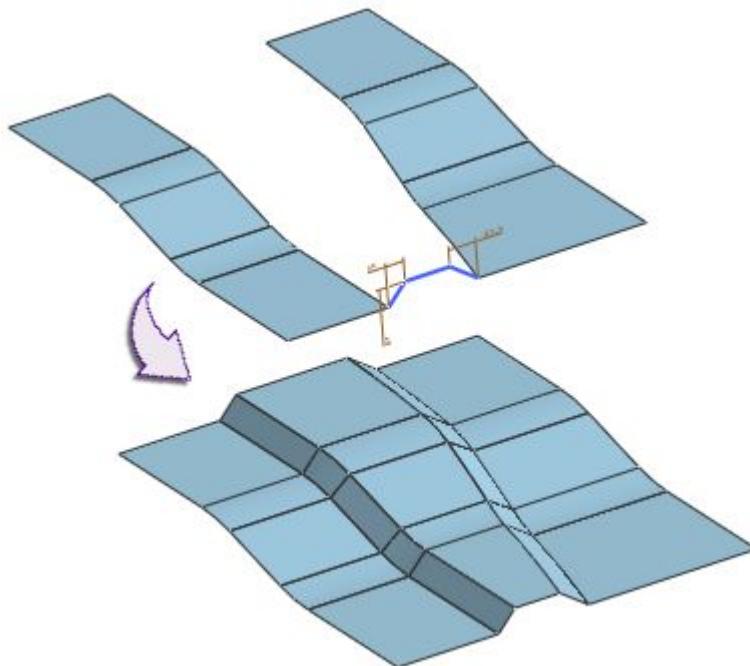


Variational Sweep overview

Use the **Variational Sweep** command to create a body by sweeping a cross section along a path where the shape of the section varies along the path.

You can do the following:

- Sweep faces that are coincident, tangent, or normal to other curves and faces.
- Add secondary sections to vary dimensions at specific locations.
- Extend the body beyond the length of the path or limit it.



Where do I find it?

Application	Modeling
Toolbar	Surface® Variational Sweep
Menu	Insert® Sweep® Variational Sweep

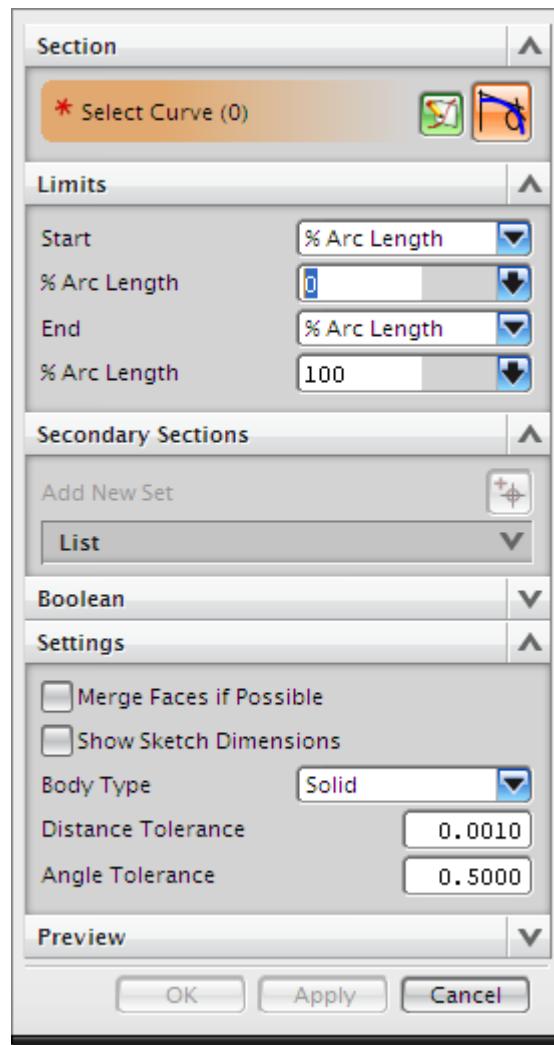


Variational Sweep dialog box

The following is an overview of the unique **Variational Sweep** creation options.

- **Limits** define the start and end locations of the sweep.
 - **% Arc Length** starts the sweep at a specified percentage.
 - **Arc Length** starts or ends the sweep at a specified length.
 - **Through Points** starts or ends the sweep at a specified point on the guide curve.
- **Secondary Sections** are copies of the primary section. You can change the dimensions, but not the shape.

- **Settings** let you define additional characteristics for the complete feature. You can merge faces, for a less complicated end result, display sketch dimensions and define the body type and tolerances.



Activities: Variational Sweep

In the *Basic Freeform* section, do the activity:

- *Create a multi-rail Variational Sweep feature*

Summary: Basic freeform

In this lesson you:

- Created a studio spline through a set of specified points.
- Created a Sketch on Path designed to define a *three dimensional* shape.
- Created a *freeform* body, a variational sweep.

Lesson

17 *Introduction to Assemblies*

Purpose

This lesson introduces the **Assemblies** application.

Objectives

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

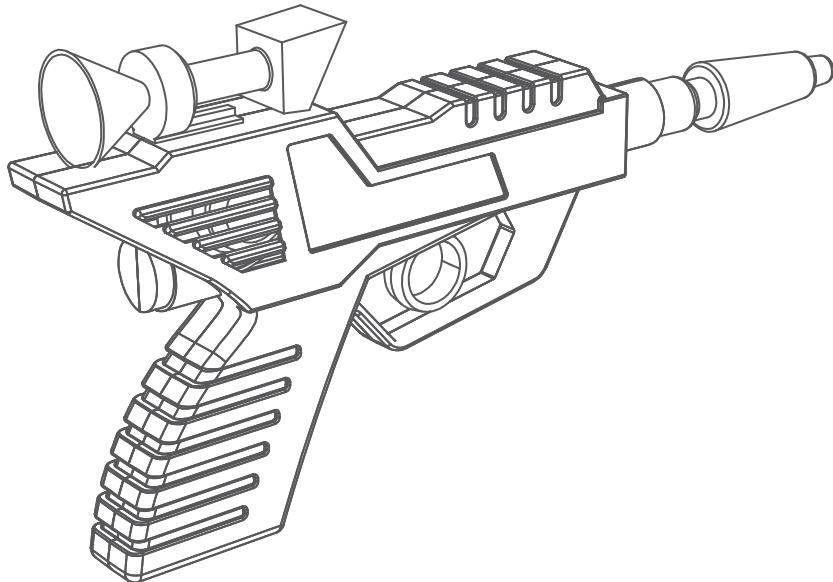
- Set load options for an assembly.
- Work with the **Assembly Navigator**.
- Check clearances between components.

Assembly

An assembly is a part which contains component objects.

Component objects are pointers to standalone parts or subassemblies.

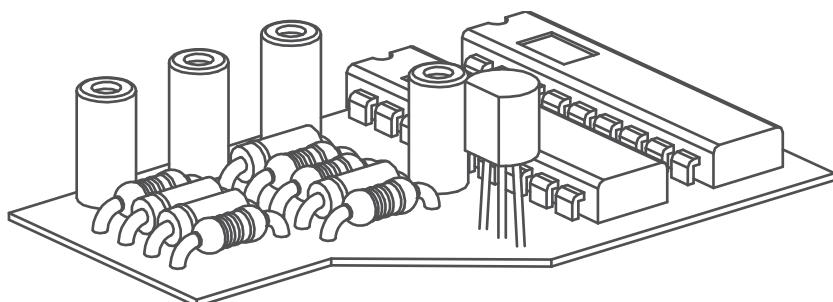
In this illustration, the toy laser gun is an assembly consisting of many components.



Subassembly

A subassembly is an assembly used as a component within a higher level assembly.

This illustration shows the subassembly of the integrated circuit board for the toy laser gun.

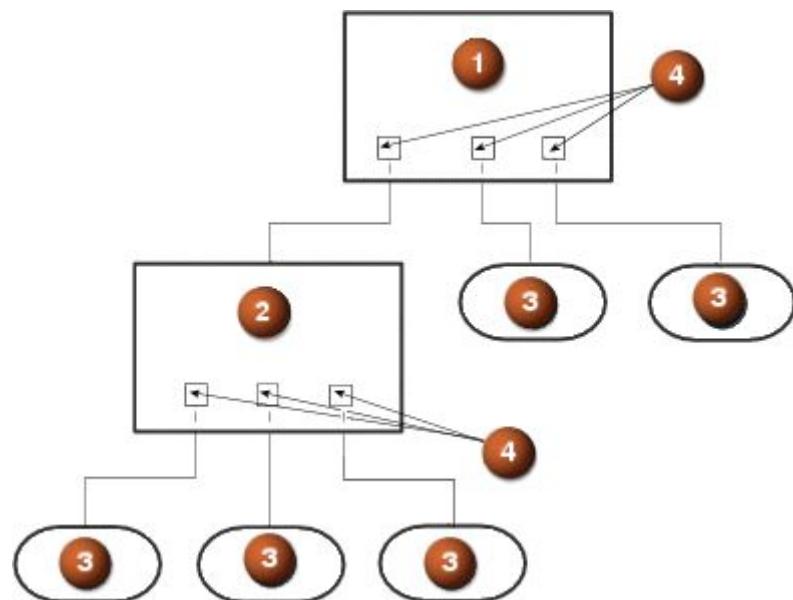


Component objects

A *component object* is a nongeometric pointer to the file that contains the component geometry. After you define a component, the part file in which you define it has a new component object. The component object allows the component to be displayed in the assembly without duplicating any geometry.

Component objects store information about the component part, such as:

- Layer.
- Color.
- Position data for the component relative to the assembly.
- The path to the component part on the file system.
- The *reference set* to display.



1. Top level assembly
2. Subassembly referenced by the top level assembly
3. Piece parts or standalone parts referenced by an assembly
4. Component objects in assembly files

Component Part Files

A *component part* is a part file which is referenced by a component object in an assembly. Geometry stored in a component part is seen, but not copied, in the assembly.

The terms *piece part* or *stand alone part* refers to a part file that is not itself an assembly.



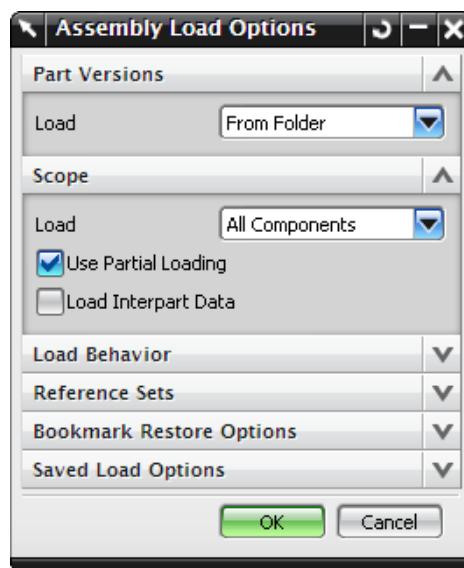
The ability to easily determine where a part file is used as a component part file is one of the strengths of Teamcenter Integration for NX.

Assembly Load Options overview

Use the **Assembly Load Options** command to configure the way components of the part you are opening are loaded into memory.

For example, you can specify:

- How to load parts with multiple versions or revisions.
- Where to find components when you work in native NX.
- Whether to load components fully or partially.
- Which default reference sets are loaded.



Where do I find it?

Menu	File® Options® Assembly Load Options
Toolbar	Standard® Open, click Options
Location in Open dialog box	Lower left corner

Part Versions group

The **Part Versions** group contains the **Load** list, with options to control how to find component parts.

- **As Saved** loads parts from the directory in which they were saved.
- **From Folder** loads parts from the same directory as the parent assembly.
- **From Search Folders** loads parts from a list of search directories.

Load states

The *load state* of a part describes the amount of data that is brought into computer memory from file storage.

There are three load states:

Fully loaded

All of the data in the file is loaded into memory. Use this load state with small assemblies, or with a subset of large assembly components to edit or create links to parametric data.

Partially loaded

Enough data is brought into memory to display the part. Parametric data is not loaded into memory. Use this load state to open large assemblies faster than when you fully open components, and to conserve computer memory for those components that you are going to modify. You can fully open partially loaded components at any time if you want to edit their data.

Not loaded

The file is not loaded into memory at all. The assembly has information about the position and the bounding box size of the unloaded parts. Use this load state to manage very large assemblies, for which you can manipulate only a relatively small amount of data at any one time. Fully or partially load only those components of immediate interest, and leave the rest unloaded.

Scope group

The **Scope** group in the **Assembly Load Options** dialog box allows you to control the assembly configuration and the *load state* of parts:

- **Load** — Control which components are opened:
 - **All Components** — Load all components.
 - **Structure Only** — Load your assembly part, but no components.
 - **As Saved** — Load the same components that were open when the assembly was last saved.
 - **Re-evaluate Last Component Group** — Load your assembly with the component group used when the assembly was last saved.
-  Component groups are advanced functionality to let you conditionally apply actions to all or part of the assembly structure.
- **Specify Component Group** — Select from a list of available component groups.
- **Use Partial Loading** — When selected, components will be partially loaded unless the **Load Interpart Data** setting requires them to be fully loaded. A partially loaded component will be fully displayed but the underlying feature data is not loaded into system memory.
- **Load Interpart Data** — Find and load parents of interpart data, even if the parts would be left unloaded by other rules.

Reference Sets

Use this area to specify a list of reference sets to be looked for, *in order*, when an assembly is loaded. The first reference set found from the top of the list reading downwards is the one that is loaded.



Think of a reference set as a subset of part geometry that you can load in place of the entire part.

The **Model** reference set is meant to contain only a body that you wish to place on a drawing.

Saved Load Options

You may save the current load options settings as your default settings. Otherwise, any changes you make in the **Assembly Load Options** dialog box apply only to your current NX session.

The **Saved Load Options** group contains options to control saved settings:

- **Save as Default** — Save the current load options as your defaults in the *load_options.def* file in your current directory.
- **Restore Default** — Reset the load options to the values defined in the *load_options.def* file in your current directory, if it exists, or to the system defaults.
- **Save to File** — Save the current load options settings to a load option definition file whose name and location you define in the **Save Load Options File** dialog box.
- **Open from File** — Open the **Restore Load Options File** dialog box, from which you can select a custom load option definitions file.



Assembly Navigator overview

The **Assembly Navigator** is a window that displays an assembly structure, component properties, and constraints between member components, in a hierarchical tree.

You can use the **Assembly Navigator** to:

- View the assembly structure of the displayed part.
- Apply commands to specific components.
- Edit the structure by dragging nodes to different parents.
- Identify components.
- Select a component.

Where do I find it?

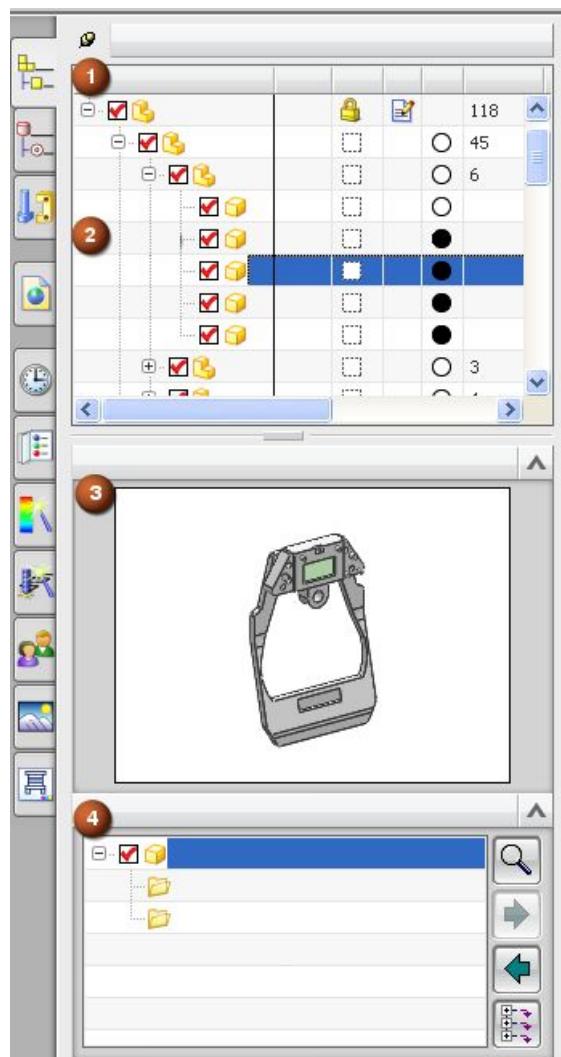
Toolbar	Resource bar ®
---------	----------------

On non-Windows platforms, the **Assembly Navigator** appears in a separate window.



Assembly Navigator user interface

1	Assembly Navigator main panel columns.	Identifies a specific component, and shows the hierachal tree.
2	Component node.	Shows information related to individual components.
3	Preview panel	Displays the saved part preview for a selected component.
4	Dependencies panel	Displays the parent-child dependencies for a selected assembly or piece part node.



Assembly Navigator hierachal tree

The hierachal tree always appears in the left-most column in the main panel of the Assembly Navigator, regardless of which column is assigned to that position.



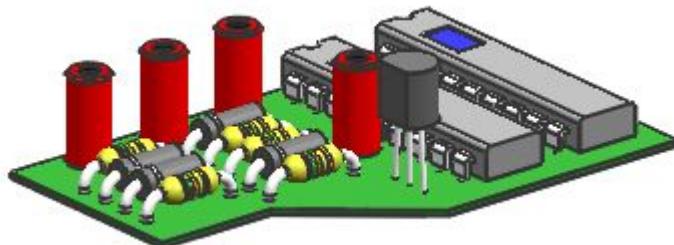
Not all available commands or options are shown in the following table.

Icon	Meaning
The following symbols precede nodes in the tree structure selected.	
	Container for view sections and component groups.
	Expands an assembly or subassembly node.
	Collapses an assembly or subassembly node.
	Indicates one or more components have been filtered out of the Assembly Navigator display. This symbol precedes the word More .
The following icons indicate that their node represents an assembly or subassembly.	
	The assembly is the work part or a component of the work part.
	The assembly is loaded, but it is neither the work part nor a component of the work part.
	The assembly is not loaded.
The following icons indicate that their node represents a piece part.	
	The component is the work part or a component of the work part.
	The component is neither the work part nor a component of the work part.
	The component is closed.
Check boxes in the tree diagram beside components	
<input type="checkbox"/>	The component is not loaded.
<input checked="" type="checkbox"/>	The component is at least partially loaded, but not visible.
<input checked="" type="checkbox"/>	The component is at least partially loaded and visible.
<input type="checkbox"/>	The part is suppressed.

Activities: Assembly load options and navigator

In the *Introduction to Assemblies* section, do the activities:

- *Assembly Load Options*



Select components

When you are using commands that require you to select components, you can select them in the following ways:

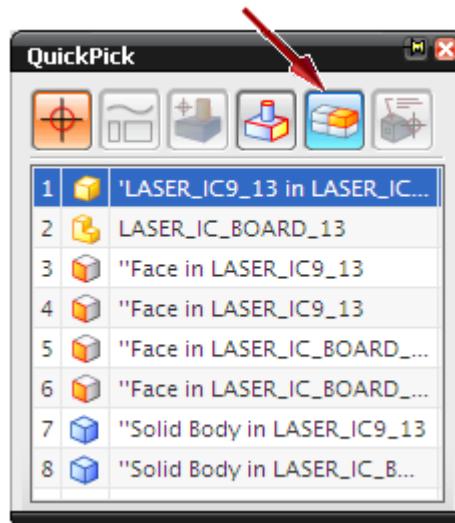
- Select the component in the graphics window.
- Select the corresponding node in the **Assembly Navigator**.
- Select the component name from a list, for certain commands.
- Type the component name in the **Class Selection** dialog box, in the **Other Selection Methods** group, in the **Select by Name** box.

Select components with QuickPick

Use the **Quickpick** list to easily select a component or object from a crowded field of objects.

Once you position your cursor over a group of components the **QuickPick** cursor displays . Click once to display the **QuickPick** dialog box.

You can use the **Components** filter in the to list only components.

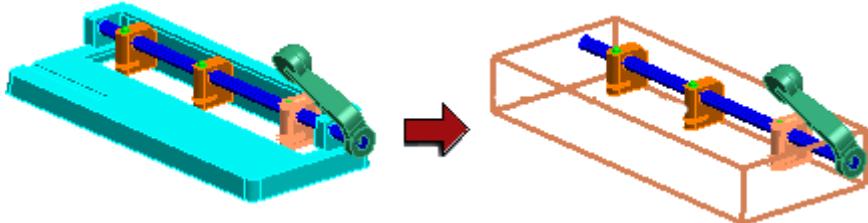




Identify components

- Select nodes for loaded and visible components in the **Assembly Navigator** hierachal tree to highlight the corresponding geometry in the graphics window.
- Move the cursor over invisible components to see a *bounding box* in the graphics window.
 - Bounding boxes are shown only when the assembly preference **Preselect Invisible Nodes** is selected.
 - If the node is packed, one random component is identified.

The following image shows how an invisible component looks when only the bounding box is displayed. The image on the left is the loaded component and the image on the right is the bounding box display.



Design in context

Design in context is the ability to create or edit component geometry while the rest of the geometry is available.

- You can change the work part while an assembly is displayed.
- There are a group of modeling and expression commands that you can use only when an assembly is displayed, to design in context.

When you are editing an assembly, you have control over the number of components that are loaded and visible. This is sometimes called the *assembly context*.

When you modify parts while you are working in the assembly context, we call it *design in context*. In other CAD applications this is occasionally called *edit in place*.

You can directly edit component geometry while the rest of the assembly is visible. The part you are editing is always the work part. The work part can be:

- A component piece part
- A component subassembly
- The displayed part

Design in context and reference sets

If you display the work part with a reference set, all geometry that you create is automatically added to the displayed reference set.



By default, the work part displays the entire part. To display a reference set for the work part, you must clear the check box for the Assemblies preference **Display as Entire Part**.



Make Displayed Part and Set Displayed Part overview

The **Make Displayed Part** and **Set Displayed Part** commands switch the display between currently loaded parts. The displayed part becomes the top node of the **Assembly Navigator** display.

If the current displayed part is a component or subassembly in an assembly, and you are going to change the displayed part to that assembly part, the current work part is maintained as the work part in the newly displayed assembly part if both of the following are true:

- You select **Preferences® Assemblies® Maintain**.
- You are in a mode where design-in-context is available.

The displayed part and the work part must always have the same units. For example, you cannot have the displayed part in inches and the work part in millimeters.

Where do I find it?

Application	Assemblies
Toolbar	Assemblies→Set Displayed Part
Menu	Assemblies→Context Control→Set Displayed Part
Graphics window	<p>Right-click a component in the graphics window® Make Displayed Part</p> <p>Right-click a component in the Assembly Navigator→Make Displayed Part</p>



Make Work Part and Set Work Part overview

Use the **Make Work Part** or **Set Work Part** command to specify the part in which to create objects and geometry. This gives you the ability to design in the context of an assembly.

Partially loaded parts

The work part is automatically fully loaded.

Work part display

You can choose **Preferences® Assemblies** and select the **Display as Entire Part** check box to show all geometry that is visible by layer settings and not hidden.

When the work part has the **Entire Part** reference set display condition you can see and use all objects in the part — such as datums, wire frame geometry, and sketches — as you continue your design in the context of the assembly.

Occurrences

When your assembly has multiple occurrences of a component, it may not be clear which occurrence becomes the work part. If you choose the name from a list, or choose a packed node in the **Assembly Navigator**, the work part can seem to be a random occurrence. If a particular occurrence is preferable to you as the work part, perhaps because it is in a convenient location, you can select its specific node in the navigator or its specific occurrence in the graphic window.

Emphasis

You can choose **Preferences® Assemblies** and select the **Emphasize** check box to emphasize the work part by changing the rest of the assembly to a predetermined color. The work part retains its assigned colors.

When you are working in a large assembly, a color difference helps to visually differentiate the work part geometry from the rest of the displayed part.

Limitations

The **Make Work Part** command is unavailable, and the **Set Work Part** command will not change the work part, if:

- The selected component has different units than the displayed part.
- The current application does not support design in context, for example, the Drafting application when a drawing is displayed.
- You have an unrelated dialog box open.

- The component is not currently visible because it is excluded by the current reference set.
- The component is not loaded.

Where do I find it?

Application	Assemblies
Toolbar	Assemblies® Make Work Part 
Menu	Assemblies® Context Control® Set Work Part
Shortcut menu	Right-click a component in the graphics window® Make Work Part Right-click a component in the Assembly Navigator® Make Work Part In the Assembly Navigator , double-click a node. In the graphics window, double-click a component with visible geometry.



Assembly Navigator display commands

Right-click a node in the **Assembly Navigator** that represents a component to display component related display commands.

The shortcut menu commands you see depend on the node you right-click.



Not all available commands or options are shown in the following table.

Command	Description
The following commands are available when you right-click a component node.	
Make Work Part	Specifies the part in which to create objects and geometry.
Make Displayed Part	Switches the display between currently loaded parts.
Display Parent	Changes the displayed part to a selected parent of the node you right-click.
Component	
Component	
Opens the selected component, according to the current assembly load option settings.	
Assembly	
Opens the selected subassembly.	
Open	
Component Fully	
Opens the selected component with full loading regardless of the Use Partial Loading setting in the assembly load options.	
Assembly	
Child Components	
Opens components of the top level assembly.	
Close	Part
	Closes the component part corresponding to the selected node.
Hide	Assembly
	Closes the entire subassembly under the selected node.
Hides the selected component.	

Show Only	Shows the selected component and hides all other components.
Delete	Removes the selected component object from the assembly, without deleting the corresponding part file. Available only for nodes with multiple occurrences.
Pack	Replaces multiple occurrences of the selected component with a single node.
Unpack	The number of occurrences represented by a packed node is displayed on the node. Available for packed nodes.



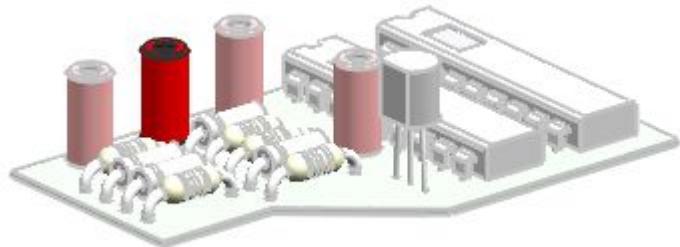
Not all available commands or options are shown in the following table.

Command	Description
The following commands are available:	
<ul style="list-style-type: none"> • In the Assembly Navigator, when you right-click the background. • On the Tools® Assembly Navigator menu. • On the Assembly Navigator toolbar. 	
	Expands all collapsed nodes in the Assembly Navigator so that every component has a visible node.
	Packs all components that currently have multiple occurrences in the assembly.
	Unpacks all components that have multiple occurrences.
Unpack All	

Activities: Assemblies — more navigator options

In the *Introduction to Assemblies* section, do the activity:

- *Additional work with the Assembly Navigator*



Part revisions and saving assemblies

After you edit it, save the work part to keep the modifications.

Use **File→Save** or **File→Save Work Part Only**.

Save

- If the work part is a standalone part, only that part is saved.
- If the work part is an assembly or subassembly, all modified component parts below it are also saved.

File→Save does not save higher level parts and assemblies if they are modified.



File→Save All saves all modified parts in the session regardless of which part is the work part, even parts that do not belong to the displayed assembly.

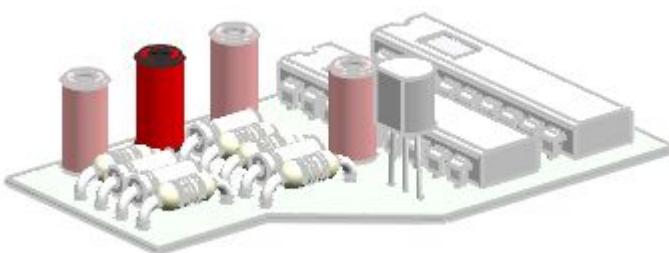


Open parts for which you do not have write privileges will not be saved.

You will get a warning about parts that cannot be saved due to permissions.

Save Work Part Only

Use the **Save Work Part Only** command to save only the work part, even if it is an assembly or subassembly with modified components.



Where do I find it?

Menu	File® Save Work Part Only
------	----------------------------------

Attributes

Attributes are numerical, time, string, or reference character strings that may be associated to an object within a part file or to the file itself.

Attributes are used to associate non-graphical information to a CAD design and display them in the following:

- **Part Navigator**
- **Assembly Navigator**
- Drawing annotation
- **Parts List**
- Graphics window

Attribute types

There are several categories of attributes.

- System attributes are globally recognized characteristics, such as color, font, width, layer, name, and group name.
- User-Defined attributes, which includes Object attributes and Part attributes

Component Properties overview

Use the **Component Properties** command to get status information about, and make changes to, selected components. Some functions can only be changed through the **Component Properties** command, including the following functions:

- Change a component name.
- Update a part family member.
- Remove color, translucency, or partial shading attributes from an occurrence, so the original component attributes are used.

There are six areas you can make changes to:

- **Assembly** controls Load Status and has options to change the layering method, control component coloring, and determine default quantities.
- **Attributes** provide the Title and Value of each existing attribute.
- **Parameters** provides **Arrangement** and **Part Family** options.
- **Weight** shows information about the component's weight and options as to when to update the Weight data.
- **Part File** provides information about the component part file such as version, creation date and time.

Where do I find it?

17

Menu	Edit→Properties
Graphics window	Right-click a component® Properties
Assembly Navigator	Right-click a component node® Properties



Assembly Navigator Properties overview

Use the **Properties** command to:

- Control the display of bounding boxes for invisible components.
- Specify the predefined columns to display.
- Define custom columns based on part attributes.
- Specify the order of columns.
- Select filtering options for components and constraints.

The first column in the **Assembly Navigator** must be one that identifies components:

- **Part Name**
- **Component Name**
- **Descriptive Part Name**
- **File Description**

Where do I find it?

	Assembly Navigator® Right-click the background® Properties
Shortcut menu	Assembly Navigator® Right-click the background® Columns® Configure



Simple Clearance Check overview

Use the **Simple Clearance Check** command to check for possible interferences between selected components and other components in the assembly.

If interferences are found, a report appears. For each interference, the report lists the following:

- The name of the selected component. An icon that shows the type of interference is displayed next to the name.
- The name of the interfering component.
- The status of the interference. For example, the report shows whether the interference is new or existing, and whether it is a hard, soft, or touching interference.

Interference Check			
Selected Component	Interfering Component	Status	Text
caster_2_fork	caster_2_spacer	New (Hard)	
caster_2_axle	caster_2_fork	New (Touching)	
caster_2_axle	caster_2_wheel	New (Touching)	
Isolate Interference			

An **Isolate Interference** option lets you view only the components involved in a selected interference. All other objects are hidden in the graphics window.

17

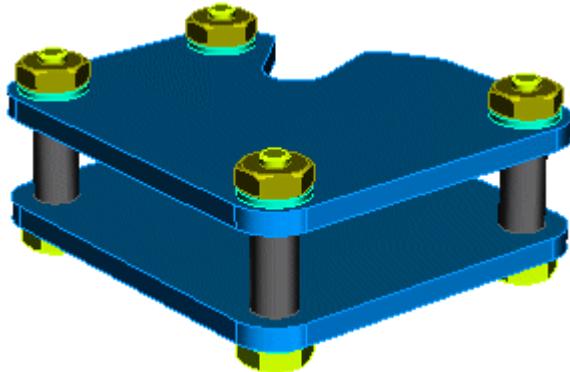
Where do I find it?

Application	Assemblies
Toolbar	Assemblies ® Assembly Clearance list ® Simple Clearance Check
Menu	Analysis ® Assembly Clearance ® Simple Clearance Check

Activities: Assembly user interface

In the *Introduction to Assemblies* section, do the activity:

- *Assembly user interface*



Summary: Assemblies

An assembly is a file which contains component objects. It is a collection of pointers to piece parts and/or subassemblies.

Assemblies provides the ability to design in context.

In this lesson you:

- Set **Assembly Load Options**.
- Worked with the **Assembly Navigator**.
- Checked clearances between components.

Lesson

18 Adding and constraining components

Purpose

This lesson introduces commands to add components to an assembly, move components, and define associative relationships between components.

Objectives

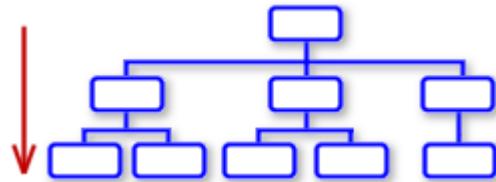
Upon completion of this lesson, you will be able to:

- Add components to an assembly.
- Move components.
- Create assembly constraints.

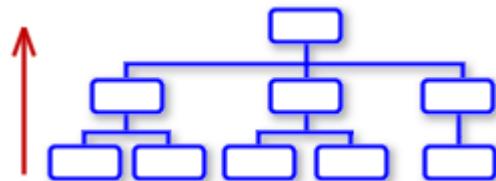
General assembly concepts

There are two approaches to creating an assembly structure.

- Top-down modeling — Create component parts at the assembly level.



- Bottom-up modeling — Create individual models in isolation, then later add them to assemblies.



You are not limited to one approach to build an assembly. For example, you can initially work in a top-down fashion, then switch back and forth between bottom-up and top-down modeling.

Bottom-up assembly modeling

In *bottom-up* assembly modeling, you create piece parts and then later add them to an assembly.

Use the **Add Component**  command to create a new component that references an existing part.

The component is added to the work part. The work part can be:

- The displayed part.
- Any member of a displayed assembly.



Add Component overview

Use the **Add Component** command to add one or more component parts to the work part.

If you add a part family template part, the **Select Family Member** dialog box appears.

You can:

- Add one or more instances of selected components in a single operation.
 - ◆ When you add multiple components at the same time, you may want to use the **Scatter** option to prevent the components from being positioned in the same location.
- Select multiple parts to add in a single operation.
- Repeat the add operation by selecting **Repeat after Add** from the **Multiple Add** list.
- Create component arrays of added components by selecting **Array after Add** from the **Multiple Add** list.

Where do I find it?

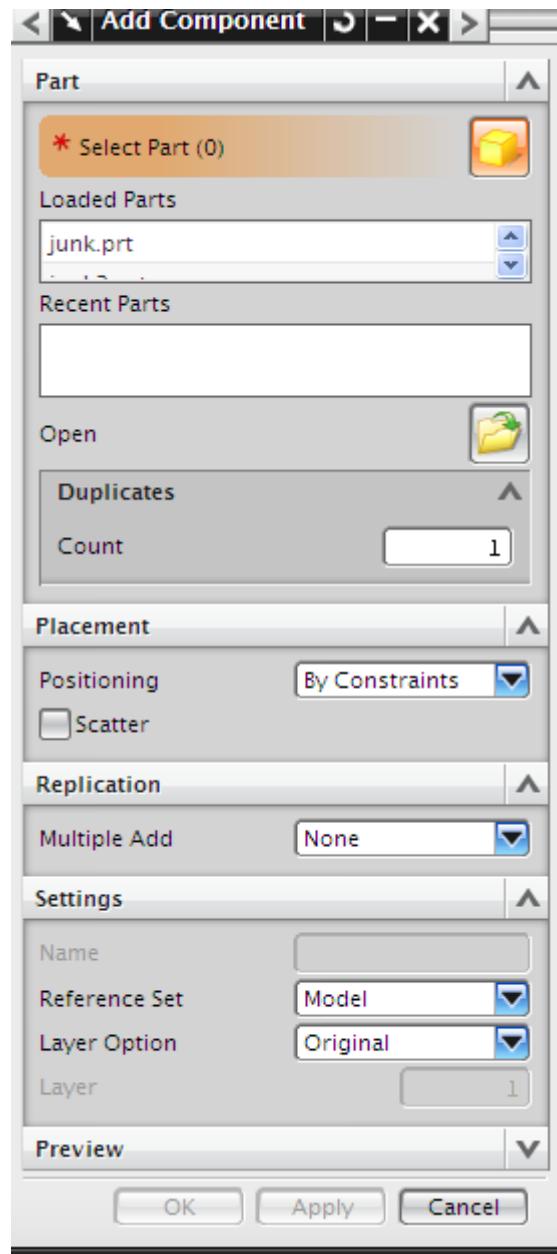
Application	Assemblies
Toolbar	Assemblies® Add Component 
Menu	Assemblies® Components® Add Component

Add Component options

The following options are found on the **Add Component** dialog box.

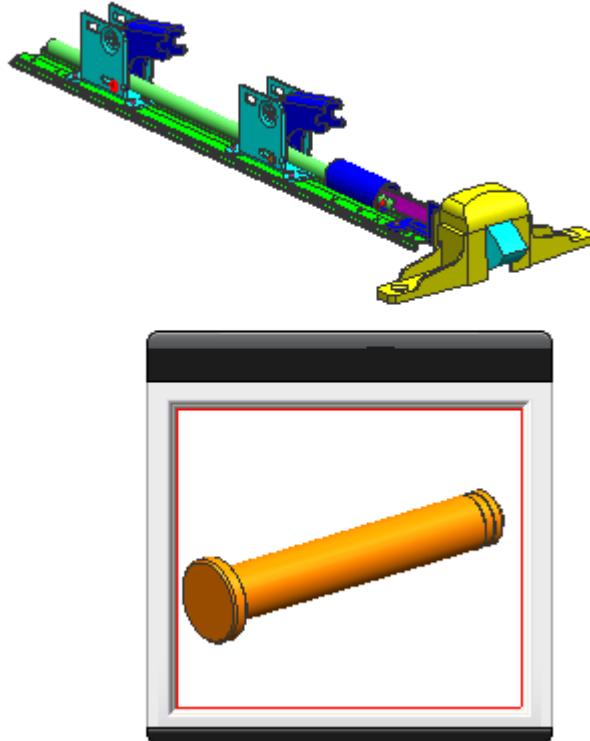
- **Part** group lets you select one or more parts to add to the work park. You can select parts from the following locations.
 - Graphics window or **Assembly Navigator**.
 - **Loaded Parts** list
 - **Recent Parts** list
- **Placement** group sets the positioning method for added components along with location options.
 - **Absolute Origin** places the components at absolute 0,0,0.
 - **Select Origin** places components at the selected location.
 - **By Constraints** opens the **Assembly Constraints** dialog box.
 - **Move Component** lets you move the component after an initial location is defined.
 - **Scatter** automatically places components in various locations to avoid component overlap.

- **Replication** group lets you use **Multiple Add** options to add more than one instance.
- **Settings** group sets component name attributes, desired reference set and layer options.



Component Preview window

In the **Assembly Preferences** dialog box, when the **Preview Component on Add** check box is selected, the **Component Preview** window appears after you select components to add to the assembly.



In the **Component Preview** staging view, you can:

- Preview the component while you position it in the assembly.
- Perform independent view operations in this window, such as zoom or rotate.
- Select geometry to define each assembly constraint.

The preview window closes at different times, depending on your selection from the **Positioning** list in the **Add Component** dialog box.

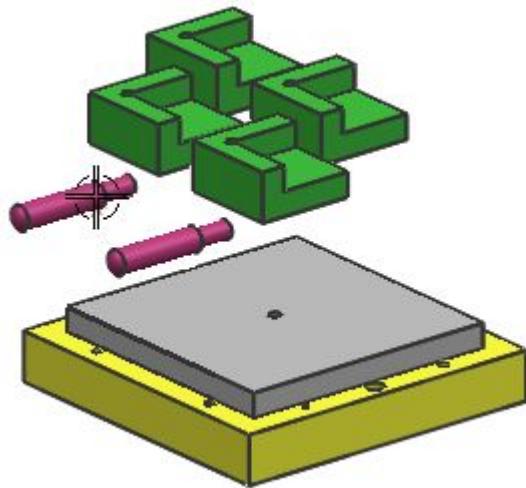
18

- **By Constraints** — The preview window closes after you specify assembly constraints to position the component.
- **Absolute Origin, Select Origin, and Move** — The preview window closes after you select the component to add and click **Apply** in the **Add Component** dialog box.

Activities: Adding components — create assembly

In the *Adding and constraining components* section, do the activity:

- *Create an assembly*





Move Component overview

Use the **Move Component** command to move and optionally copy a component in an assembly. You can select and move multiple components if they have the same parent.

You can move components dynamically or you can create constraints to move the components into position.

You can move components in your work part by default.

If you want to move components anywhere in your assembly, regardless of what your work part is; for example, if your work part is often a subassembly, you can change the **Move Component Scope** assembly positioning customer default from **Work Part** to **Anywhere in Assembly** and restart NX.



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

Where do I find it?

Application	Assemblies
Toolbar	Assemblies® Move Component
Menu	Assemblies® Component Position® Move Component

Move Component options

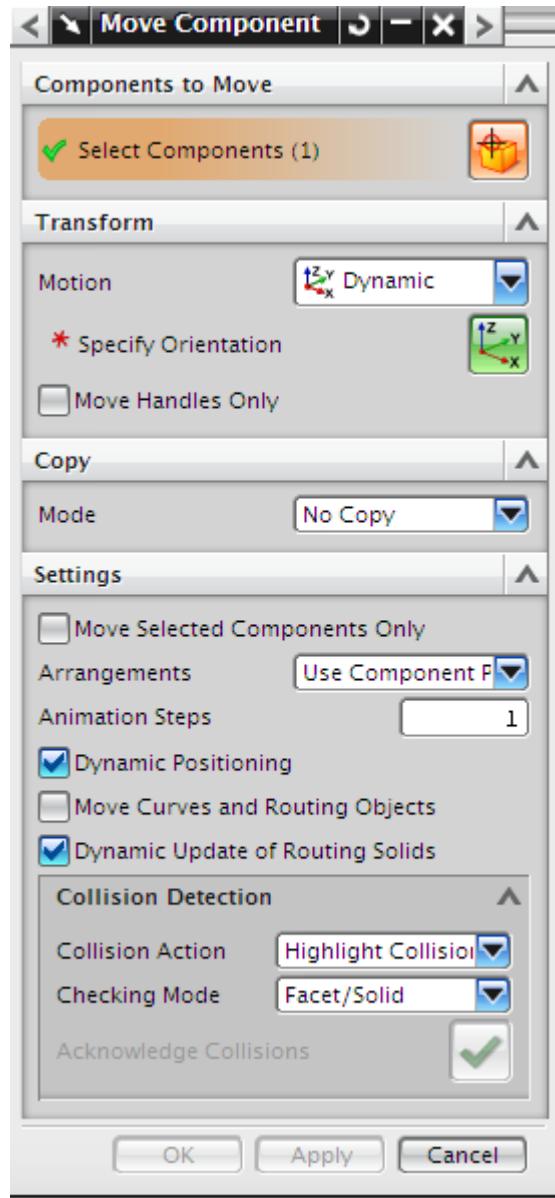
The following options are found on the **Add Component** dialog box.

- **Transform** group specifies how the selected components move.
Motion provides the following options to move a component.
 - **Dynamic** use drag functions, the on-screen input box or the **Point** dialog box.
 - **By Constraints** lets you add assembly constraints.
 - **Distance** moves the component a distance from a defined point in a specified direction.
 - **Point to Point** moves the component from one point to another.
 - **Delta XYZ** enter a distance based on the displayed parts absolute or the WCS location.
 - **Angle** moves the component about an axis point and vector direction a specified angle.
 - **Rotate By Three Points** defines the pivot point, start and end points in which to move the component.
 - **CSYS to CSYS** move the component from one CSYS to another.
 - **Axis to Vector** move the component between two defined vectors about a pivot point.
- **Copy** group lets you copy components that are moved.

- **Settings** group lets you determine how moved components will behave within arrangements, how many animations steps are used when the component is moved, options for routing objects, and collision detection.



Collision Detection is only available when **Copy** is set to **No Copy**.



Assembly Constraints overview

Use the **Assembly Constraints** command to define positions of components in the assembly. NX uses directionless positioning constraints, which means that either component can move to solve the constraint.

You can use assembly constraints to:

- Constrain components so they touch each other or align with each other. The Touch Align constraint is the most commonly-used constraint.
- Specify that a component is fixed in place. This is useful when you want to control which component moves when the software solves a constraint.
- Bond two or more components together, so they move together.
- Define a minimum distance between selected objects in components.

See Assembly constraint types for more information about the different types of constraints and their uses.

You can convert mating conditions to assembly constraints. Assembly constraints are usually faster to create and easier to use than mating conditions.

You can delay the updating of assembly constraints until a convenient time. When you are ready, you can activate the update.

You can temporarily display the degrees of freedom for a selected component.

Where do I find it?

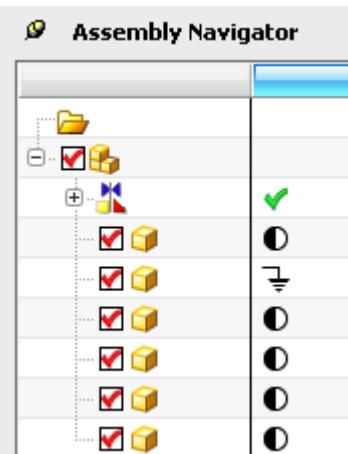
Application	Assemblies
Prerequisite	Set Preferences® Assemblies® Interaction to Assembly Constraints .
Toolbar	Assemblies® Assembly Constraints 
Menu	Assemblies® Component Position® Assembly Constraints

Assembly constraint types

Assembly constraint type	Description
 Angle	Defines an angle dimension between two objects.
 Bond	<p>“Welds” components together so they move as a rigid body.</p>  Bond constraints can only be applied to components, or to components and assembly-level geometry. Other objects are not selectable.
 Center	Centers one or two objects between a pair of objects, or centers a pair of objects along another object.
 Concentric	Constrains circular or elliptical edges of two components so the centers are coincident and the planes of the edges are coplanar.
 Distance	Specifies the minimum 3D distance between two objects.
 Fit	<p>Brings together two cylindrical faces with equal radii. This constraint is useful for locating pins or bolts in holes.</p> <p>If the radii later become non-equal, the constraint is invalid.</p>
 Fix	<p>Fixes a component at its current position.</p>  A fix constraint is useful when you need an implied stationary object. With no fixed node, the entire assembly has freedom to move.
 Parallel	Defines the direction vectors of two objects as parallel to each other.
 Perpendicular	Defines the direction vectors of two objects as perpendicular to each other.
 Touch Align	<p>Constrains two components so they touch or align with each other.</p>  Touch Align is the most commonly-used constraint.

Assembly constraints and the Assembly Navigator

The positional constraints of a component are reflected as symbols in the **Position** column in the Assembly Navigator.



This list shows examples of the most used icons, not all possible constraint symbols.

● Fully constrained	All six degrees of freedom are constrained.
⊖ Fixed	There are no remaining degrees of freedom due to the fixed constraint.
○ Partially constrained	The component has at least one remaining degree of freedom.
✗ Inconsistently constrained	Two or more constraints conflict.
❓ Deferred constraints	There are constraints that reference unloaded data, so the position may be subject to change.
○ Unconstrained	The component has all six degrees of freedom and no position overrides.
○ Suppressed	All constraints for the component are suppressed.

For **Constraints** node, one of the following icons may appear:

- ✓ **All Geometry Loaded**
- ❓ **Some Geometry Unloaded**

Create a Touch Align constraint

A touch align constraint constrains two components so they touch or align with each other. This is the most common constraint.

1. On the **Assemblies** toolbar, click **Assembly Constraints** 
2. In the **Assembly Constraints** dialog box, set **Type** to **Touch Align**.
3. Check the **Settings** and modify them if you do not want to use their defaults:
4. Set **Orientation** to one of the following:
 - **Prefer Touch** presents a touch constraint when touch and align constraints are both possible.
 - **Touch** constrains objects so their surface normals are in opposite directions.
 - **Align** constrains objects so their surface normals are in the same direction.
 - **Infer Center/Axis** specifies that when you select a cylindrical or conical face, the face center or axis is for alignment.
5. Click **Select Two Objects**  (if necessary), and select two objects for the constraint.
You can use the **Point Constructor**  to help you select objects.
6. If two solutions are possible, you can click **Reverse Last Constraint**  to flip between the possible solutions.
7. Click **OK** or **Apply**.

Create a Center constraint

A center constraint centers one or two objects between a pair of objects, or centers a pair of objects along another object.

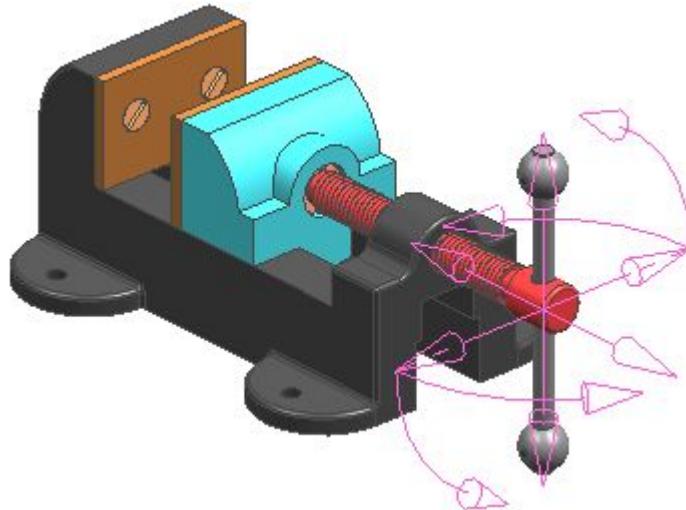
1. On the **Assemblies** toolbar, click **Assembly Constraints** .
2. In the **Assembly Constraints** dialog box, set **Type** to **Center**.
3. Check the **Settings** and modify them if you do not want to use their defaults:
4. Specify the **Subtype**:
 - **1 to 2** centers the first selected object between the next two selected objects.
 - **2 to 1** centers two selected objects along the third selected object.
 - **2 to 2** centers two selected objects between two other selected objects.
5. If **Subtype** is **1 to 2** or **2 to 1**, set **Axial Geometry** to define what happens if you select a cylindrical face or circular edge:
 - **Use Geometry** uses selected cylindrical faces for the constraint.
 - **Infer Center/Axis** uses the center or axis of the object.
6. Click **Select Objects**  (if necessary), and select the appropriate number of objects as defined by the **Subtype**.
 You can use the **Point Constructor**  to help you select objects.
7. If two solutions are possible, click **Reverse Last Constraint**  to flip between the possible solutions.
8. Click **OK** or **Apply**.



Show Degrees of Freedom overview

Use the **Show Degrees of Freedom** command to temporarily display the degrees of freedom for a selected component.

Degrees-of-freedom arrows appear in the graphics window, and the Status line shows the number of rotational and translational degrees of freedom that exist in the component.



You can find degrees of freedom on components that are loaded and unsuppressed. If geometry in other components needs to be loaded to find the degrees of freedom, you receive a message asking if you want to load the geometry.

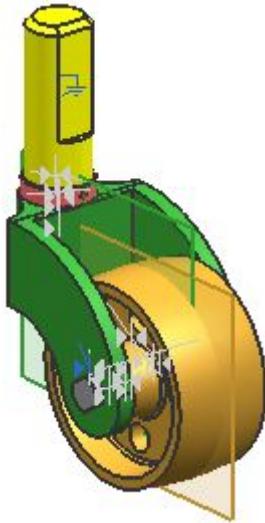
Where do I find it?

Application	Assemblies
Toolbar	Assemblies® Component Position Drop-down list® Show Degrees of Freedom
Menu	Assemblies® Component Position® Show Degrees of Freedom
Graphics window	Right-click a component® Show Degrees of Freedom
Assembly Navigator	Right-click a component node® Show Degrees of Freedom

Activities: Constrain and move components

In the *Adding and constraining components* section, do the following activity:

- *Constrain and move components*



Summary: Adding and constraining components

When you add components to an assembly, you reference other part files or subassemblies.

You can move components or establish constraints to define the locations of components.

In this lesson you:

- Added components to an assembly.
- Moved components.
- Defined assembly constraints.

Lesson

19 Reference Sets

Purpose

Reference sets allow you to limit the amount of component part information displayed in an assembly. Reference sets will also allow you to show alternate representations or simplified versions of the model.

Objectives

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

- Create reference sets
- Replace reference sets

Reference sets overview

Use **Reference Set** commands and options to control the display of a component or subassembly part in higher level assemblies. Reference sets are a named collection of objects in a piece part or subassembly.

There are two types of reference sets:

- Automatic reference sets that are managed by NX.
- User-defined reference sets.

There are two primary reasons to use reference sets:

- Filter unwanted objects in a component part so that they do not appear in the assembly.
- Represent a component part in the assembly with alternative or simpler geometry than the complete solid body.

A well-managed reference set strategy can lead to:

- Faster load times.
- Reduced memory usage.
- Less cluttered graphics displays.

You can load any reference set lightweight, which means that lightweight representations display for the members of the reference set that have lightweight representations. Set the **Automatic Lightweight Generation** customer default to specify the reference sets where the software automatically maintains lightweight representations.



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

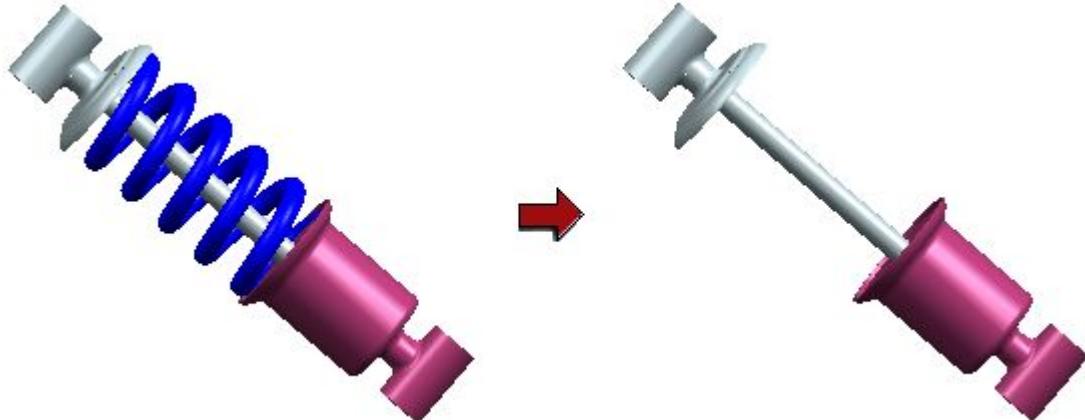
Where do I find it?

Menu	Format→Reference Sets
------	------------------------------

Default reference sets

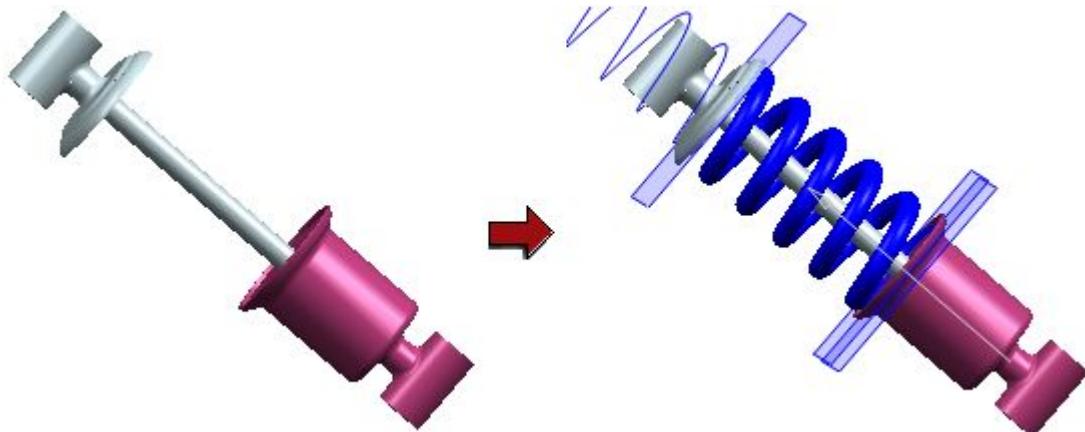
Every component or subassembly owns two reference set display conditions that exist in every part file:

- The **Empty** reference set displays nothing in the graphics window.



If a component is represented by the empty reference set it is considered to be an excluded reference set. The check box for the corresponding node in the Assembly Navigator is unavailable. You must replace the empty reference set with a valid reference set in order to view the contents.

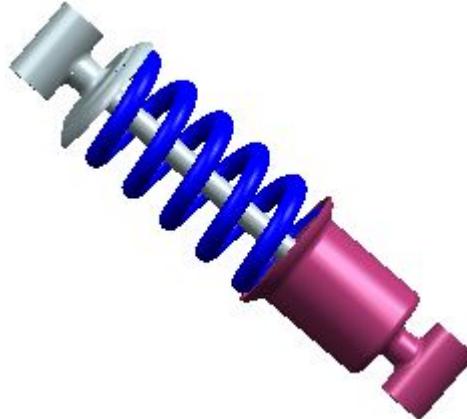
- The **Entire Part** reference set displays all of the component objects in the graphics window.



Automatic default reference sets

The following reference sets can be created as you work in your component and subassembly part files:

- The **Model** reference set displays only the completed model.

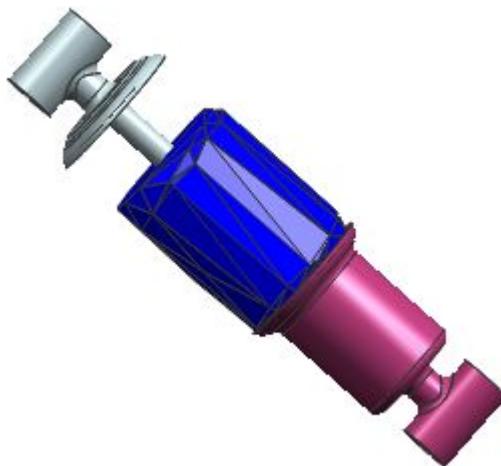


- The **Simplified** reference set is maintained automatically if you define a name for it in customer defaults.

Once you have defined a simplified reference set, any wrap assembly or linked exteriors that you create are automatically added to it.



The following example displays the simplified reference set with a wrapped component.



To define reference set default options go to **File**→**Utilities**→ **Customer Defaults**→**Assemblies**→**Site Standards**.

User-defined reference sets

The default reference sets generated by NX do not always suit your design criteria. You can define your own reference sets to make sure your assembly display meets your needs.

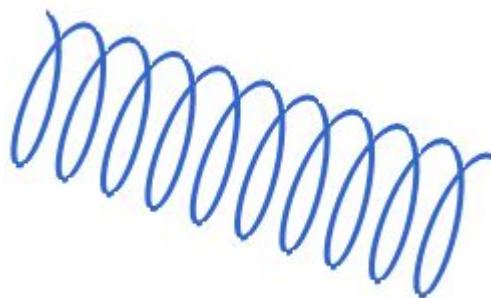
The following is a list of scenarios where you may find it beneficial to create user-defined reference sets:

- You use datums as reference features to apply assembly constraints. Instead of using the entire part reference set create a user-defined reference set named **MATE** and add only the datums.
- Your complex assembly contains multiple occurrences of a standard part, such as fasteners, and you need to improve your computer performance. Create a user-defined reference set named **Simple** and use only the centerline and a profile curve for the display.



Wireframe reference set displays are selectable and are not affected by hidden line removal settings.

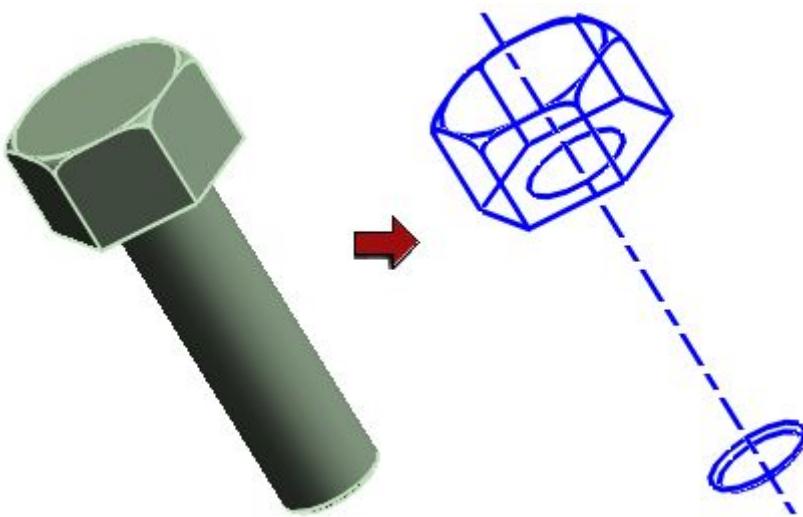
- You need to display theoretical intersections or centerlines so you can dimension to them on a drawing. Create a user-defined reference sets named **Draft** and add the necessary curves.



Create a new reference set

This example shows you how to create a user-defined reference set that simplifies the display of a standard part and improves performance.

1. In the **Assembly Navigator**, right-click the component or subassembly that will own the reference set and choose **Make Displayed Part**.
2. Prepare your part by extracting the necessary curves, points, or sheets that you want displayed in your reference set.
3. Choose **Format→Reference Sets**.
4. In the **Reference Sets** dialog box, click **Add New Reference Set** .
5. In the **Reference Set Name** box, type **Simple**.
6. (Optional) In the **Settings** group, select the **Add Components Automatically** check box to automatically add new components to the reference set as you create them.
7. In the graphics window, select objects until all the objects you want in the reference set are included.
8. Click **Close**.



Edit a reference set



Empty and entire part references sets are not available for edit.

1. In the **Assembly Navigator**, right-click the component or subassembly that owns the reference set and choose **Make Work Part**.
2. Choose **Format→Reference Sets**.
3. In the **Reference Set** dialog box, from the Reference Set list box, select the **Reference Set** you want to edit.
4. You can perform the following edits on a selected reference set.
 - Add or remove objects.
 - Rename the reference set.
 - Adjust the **Add Components Automatically** option for user-defined reference sets.

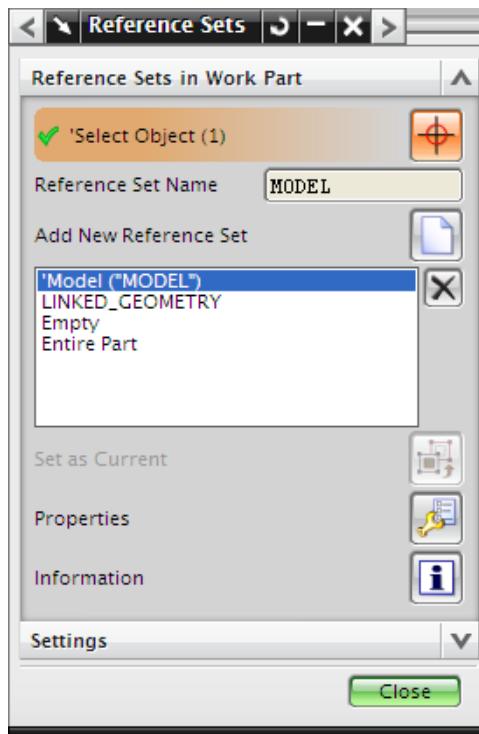
Reference Set information

When you request information on a reference set using the **Information→Assemblies→Reference Set** method, the system will:

- Select the members of the set in the graphics window.
- Display the origin and orientation in the graphics window.
- Provide a listing of relevant data in the Information window.

When you request information on a reference set by clicking **Information** from the **Reference Sets** dialog box, the system will:

- Provide a listing of relevant data in the Information window.



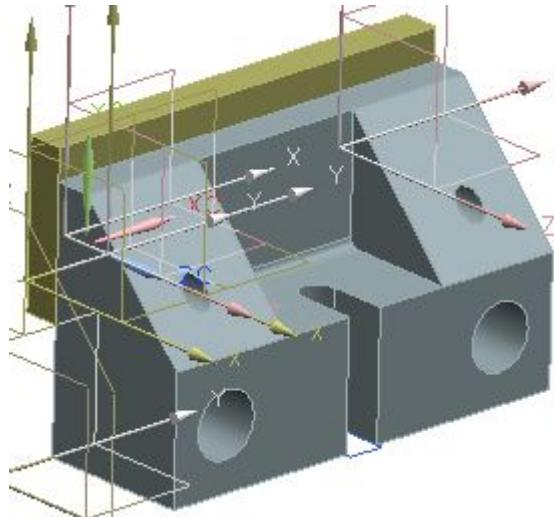
You can obtain information about reference sets that reside in the work part:

- Choose **Information→Assemblies→Reference Set**
- In the **Reference Sets** dialog box, click **Information**.

Activities: Create and examine reference sets

In the *Reference Sets* section, do the following activities:

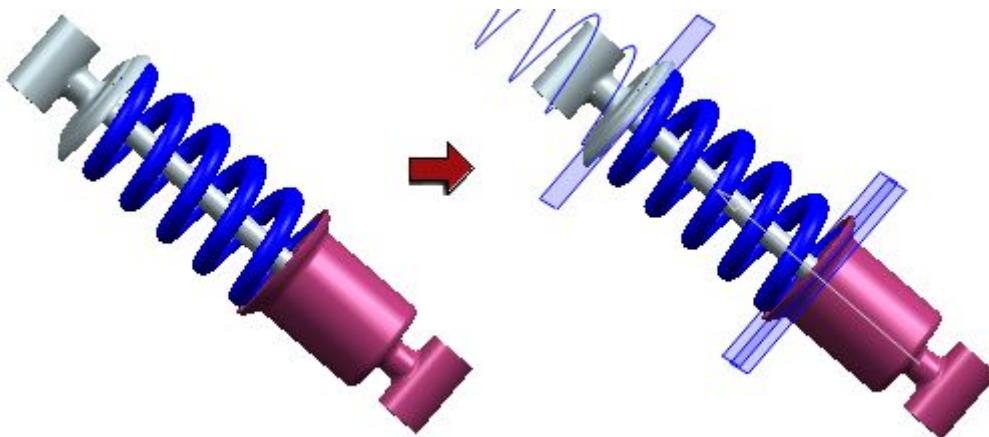
- *Create reference sets*





Replace Reference Set overview

Use the **Replace Reference Set** command to switch the component display and manage your assembly graphics window.



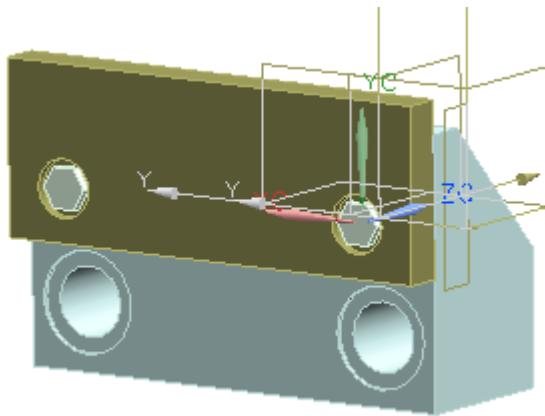
Where do I find it?

Prerequisite	Work in the context of an Assembly.
Toolbar	Assemblies → Replace Reference Set
Menu	Assemblies → Components → Replace Reference Set
Graphics window	Right-click on one or more components in the Assembly Navigator and choose Replace Reference Set .

Activities: Replace Reference Sets in an assembly

In the *Reference Sets* section, do the activity:

- *Replace reference sets in an assembly*



Summary

Reference sets are used to limit the amount of information referenced by the component object in an assembly or subassembly. They allow you to create different displays of the same assembly or component to simplify the assembly or provide alternate configurations.

In this lesson you:

- Defined a hierarchy of reference sets to be loaded using Assembly Load Options.
- Added reference sets.
- Replaced reference sets.

Lesson

20 Top-down assemblies

Purpose

Creating data at the assembly level is a common practice and typically referred to as *top-down* assembly modeling.

Objectives

In this introductory lesson you will learn how to:

- The top-down assembly creation methods.
- Modeling and sketching in the context of an assembly.

Top-down assembly modeling

With the *top-down* assembly modeling, you can create geometry at the assembly level, and move or copy it to one or more components.

As you select and transfer geometry, you can automatically transfer objects such as datums that were used to define positions but not directly selected. All associative relationships are maintained. So if you select a sketch that resides on a datum CSYS the datum CSYS is also be transferred over to the new part file.

Use the **Create New Component**  command to create new part files.

For geometry selection, you can do the following:

- Select geometry that is currently at the assembly level , and move it to the new component.
- Select geometry to copy to the new component. Do this if, for example, you are going to export the same geometry to other new components.
- Select no geometry, to create a new instance of an existing template or a blank part.



Create New Component overview

Use the **Create New Component** command to create a component part file and reference it in the assembly work part. When you create a component, feature parameters are maintained.

You can use **Create New Component** to create an assembly using a top-down design method.

With the top-down method, you can design a:

- Copy or move existing geometry into a new component.
- Create an empty component and add geometry to it later.

NX creates a new component part containing the selected geometry and also creates a component in the assembly work part.



You must save the new component. It is not saved automatically.

Where do I find it?

Application	Assemblies
Toolbar	Assemblies® Create New Component 
Menu	Assemblies® Components® Create New Component



Create a new component

1. On the **Assemblies** toolbar, click **Create New Component** or choose **Assemblies® Components® Create New Component**.
2. In the **New Component File** dialog box, select a template.
3. (Optional) Change the default name and folder location if desired and click **OK**.
4. (Optional) Select objects in the graphics window, and then click **OK** to add existing objects to the new component.
5. (Optional) In the **Create New Component** dialog box, in the **Settings** group, modify any of the following:
 - In the **Component Name** box, enter the name for the new component.
 - Specify a reference set for any geometry you are copying or moving.
 - From the **Layer Option** list, select the layer on which to display component geometry.
 - From the **Component Origin** list, select **WCS** or **Absolute**.

This specifies whether you want the component origin aligned with the parent assembly's WCS or the absolute coordinate system.
 - Select or clear the **Delete Original Objects** check box to specify whether you want to delete any original geometry selected from the assembly.

Deleting the original geometry is similar to a move operation, and retaining the geometry is similar to a copy operation.
6. Click **OK**.
7. In the **Assembly Navigator** check to ensure that your new component is added to the assembly structure in the proper location.

Verify the creation of a new component

When you create a new component, it may not be obvious that the component was created.

There are a few ways to verify the creation of a new component:

- Locate the new component in the **Assembly Navigator**.
- **Information→Assemblies→List Components** lists all components in the assembly.
- Change the work part to the new component with **Assemblies→Context Control→Set Work Part**.
- **Status line** briefly displays a new component creation message.

Data selection during component creation

Adding data to a new component can be thought of in terms of moving or copying the data into the new part. If **Delete Original Objects** is selected, data is moved; otherwise it is copied.

- All geometry, whether moved or copied, will have the same color and show/hide-status as the original. The occurrences of that geometry created in the assembly will "look" identical to the originals.
- If you attempt to "move" an object, and some other object which you are not moving depends on that object, then the selected object will in fact be "copied".



If you select a sketch (which has been extruded) to be moved to a component, but you do not select the associated extruded body, the sketch will be copied.

If you select a line which is part of a sketch to be moved to the component, but you do not select the sketch, the line will be copied.

- If you copy only a sketch and the sketch has a swept solid associated with it, the copied sketch will not be associated to the solid. If the sketch is attached to a face, the body it is attached to will be copied.
- If you move a solid that was created from a sketch, the sketch is copied.
- Any expressions that the sketch uses are copied into the new part. Any expressions that are not required by the sketch are not copied.

Design in context

Design in context is the ability to create or edit component geometry while the rest of the geometry is available.

- You can change the work part while an assembly is displayed.
- There are a group of modeling and expression commands that you can use only when an assembly is displayed, to design in context.

When you are editing an assembly, you have control over the number of components that are loaded and visible. This is sometimes called the *assembly context*.

When you modify parts while you are working in the assembly context, we call it *design in context*. In other CAD applications this is occasionally called *edit in place*.

You can directly edit component geometry while the rest of the assembly is visible. The part you are editing is always the work part. The work part can be:

- A component piece part
- A component subassembly
- The displayed part

Design in context and reference sets

If you display the work part with a reference set, all geometry that you create is automatically added to the displayed reference set.



By default, the work part displays the entire part. To display a reference set for the work part, you must clear the check box for the Assemblies preference **Display as Entire Part**.

Model in context

Many Modeling commands let you select geometry directly from other components.

When you use these commands, you can automatically copy selected geometry into your work part as WAVE-linked associative geometry, or as nonassociative geometry, depending on your Selection bar settings.

Objects you can copy with the **WAVE Geometry Linker** include:

- Edges
- Points
- Faces
- Bodies
- Datums

You can use copied objects to:

- Create geometry
- Edit geometry
- Constrain geometry

Sketching in context

Some Sketch commands allow you to select geometry from any component in the assembly, but do *not* create associative interpart links. Examples of such commands are:

- **Profile**
- **Line**
- **Arc**
- **Circle**
- **Derived Lines**

You can use the **Selection Scope** option **Entire Assembly** when you want the sketch to snap to existing geometry on other parts in the assembly.

Some Sketch commands allow you to create interpart links. Such commands include:

- **Project Curve**
- **Intersection Point**
- **Intersection Curve**

Design in context selection scope

The **Selection Scope** option on the Selection bar helps you design in the context of an assembly.



Use the **Selection Scope** option to indicate the selection range you want.

You can select objects from the:

- Entire assembly
- Current work part only
- Work part and its components

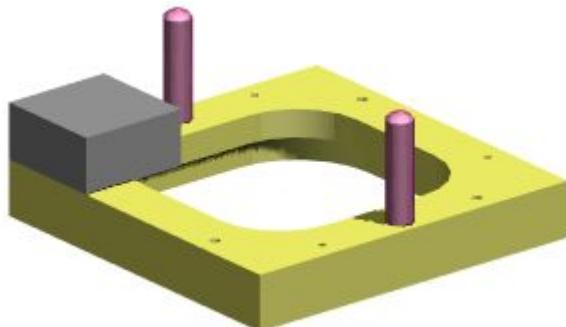
Some reasons to select geometry in parts other than the work part are to:

- Select faces or edges from adjacent components to define the profile of an extruded or revolved feature in the work part.
- Select a face to define the limit of an extrusion in the work part.

Activities: Top-down assembly modeling

In the *Top-down assembly modeling* section, do the activity:

- *Top-down assembly modeling*



Summary: Top-down assembly modeling

Top-Down Assembly Modeling allows you to build new components in relation to other components within the same assembly.

In this lesson you:

- Created new components using the Top-Down method.
- Designed in Context of the assembly.

Lesson

21 Assembly Arrangements

Purpose

The lesson introduces the concepts of using assembly arrangements to specify alternative positions for one or more components in your part.

Objectives

On completion of this lesson you will be able to:

- Set the active and default arrangement.
- Use arrangement properties to ignore assembly constraints and properly reposition a variation of your vise assembly.



Assembly Arrangements overview

Use the **Assembly Arrangements** command to define alternative positions for one or more components or subassemblies in your part.

You can either move or suppress components to create an alternate arrangement.

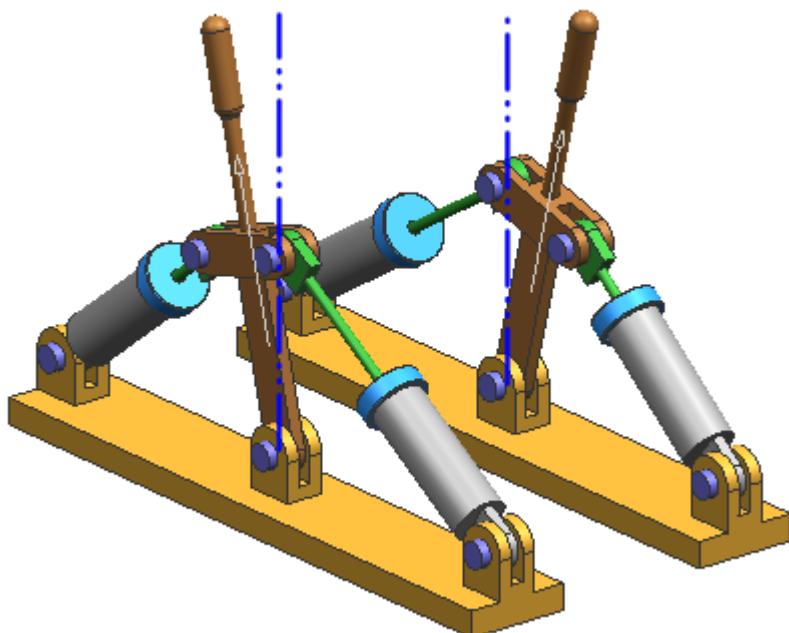
Arrangements are created in assemblies or subassemblies and must have one active arrangement and one default arrangement (or one arrangement that is both active and default). You can create as many arrangements as you want.

An assembly Arrangement determines:

- The position and orientation of the immediate child components.
- The variable component positioning of any subcomponents.
- The used assembly arrangement for each immediate child component.
- The arrangement specific suppression of components and subassemblies.
- The arrangement specific assembly constraints.



In the following example, two identical subassemblies display two different arrangements. The arrangement on the left displays the lever in the back position the arrangement on the right displays the lever in the forward position.



Where do I find it?

Application	Assemblies
Toolbar	Assemblies→Assembly Arrangements 
Menu	Assemblies→Arrangements

Assembly Arrangement status

Assembly arrangements are assigned to one of the following status:

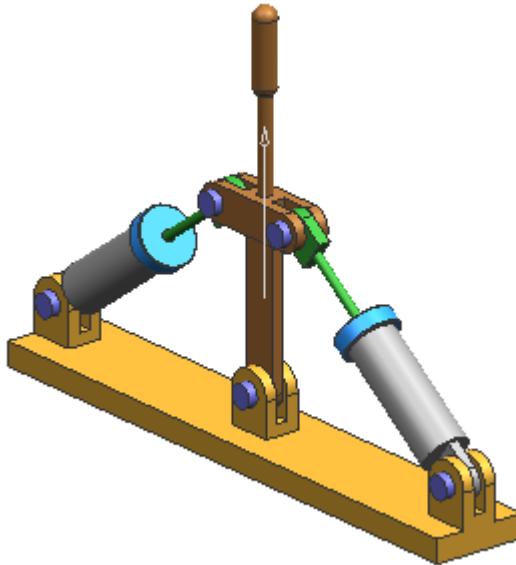
- Active arrangements:
 - Active arrangements are in effect for the displayed part.
 - Control (directly or indirectly) the arrangements used by every subassembly below the displayed part.
 - Are marked with a green check in the list box of the **Assembly Arrangements** dialog box.
- Default arrangements:
 - Are not necessarily the active arrangement, you can display an inactive default arrangement.
 - Are used by external applications like Teamcenter Integration that need positioning information about components.
 - Are created automatically when you add or create the first component in a new part, or load a pre-NX2 assembly.
 - Can be reassigned in the **Assembly Arrangements** dialog box.
- Used arrangements:
 - Are in subassemblies of the displayed part that controls a subassembly.
 - Are determined by the arrangement of its parent.
 - Display in the **Assembly Navigator Arrangement** column for each subassembly (if the column is blank, only one arrangement is available).
 - Are marked with a green check in the list box of the **Assembly Arrangements** dialog box.



Create an Assembly Arrangement

Use this procedure to create **Assembly Arrangements**.

1. In the **Assembly Navigator**, select an assembly or subassembly.



2. On the **Assemblies** toolbar, click **Assembly Arrangement** , or choose **Assemblies→Arrangements**.

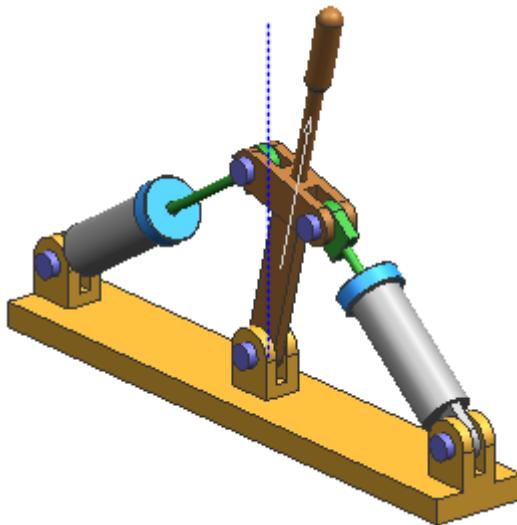
3. From the list box, select an arrangement and click **Copy** .
4. Rename the copied arrangement.

 You can accept the new default name, rename it immediately, or rename it at a later time with the **Rename** option .

5. Move the components to the new positions within the assembly and save your parts.

 Check for arrangement specific options in either the **Move** or **Supression** dialog boxes.

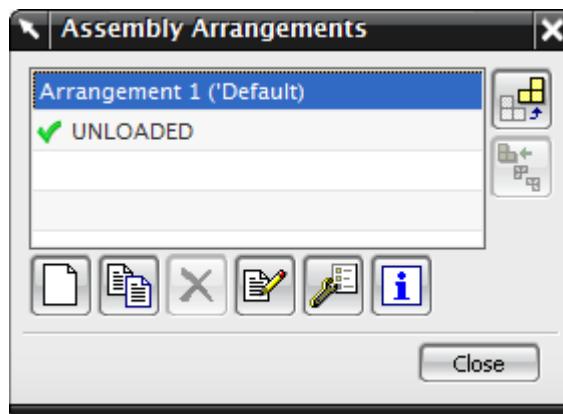
 Once you have one or more arrangements, you can switch arrangements by selecting the one that you want from the **Assembly Navigator**→**Arrangements** shortcut menu



6. (Optional) To make the new **Arrangement** the default, click **Use** .

Assembly Arrangements dialog box

-  **Use** is available when multiple arrangements exist. Sets the selected arrangement as the current arrangement being used.
-  **Set as Default** is available when multiple arrangements exist. Sets the selected arrangement as the default.
-  **New Arrangement** creates a new arrangement.
-  **Copy** copies an existing arrangement, which you can then modify and rename to create a new arrangement.
-  **Delete** is available when multiple arrangements exist. Deletes the selected arrangement.
-  **Rename** lets you rename a selected arrangement.
-  **Properties** opens the **Arrangement Properties** dialog box where you can edit the name along with other options.
-  **Information** displays information about the selected arrangement.



Assembly Arrangement notes

When you create or edit an **Assembly Arrangement**, consider the following:

- You cannot delete an arrangement while it is the default. If you delete the active arrangement, the default arrangement becomes the active arrangement.
- Assembly arrangements do not directly control the positions of geometry, only of components. Control of geometry can occur when it depends on component positioning.
- Occurrences of the same subassembly can use different arrangements.
- Unless you select **Ignore All Constraints** in the **Properties** dialog box, assembly constraints are solved in the used arrangement.
- A component can have different suppression states in different arrangements in the same assembly.

Use the **Suppression** dialog box to specify any arrangement-specific suppression states.

- Exploded views are arrangement-specific. This lets you show two arrangements on the same drawing.
- Wave links are arrangement-specific. NX remembers the arrangement of the lowest common ancestor when a WAVE link is created and positions the geometry accordingly.
- Measurement features update with arrangement changes only when the measurement expression is in a different component than the one where the arrangement change occurs.



An assembly contains a subassembly and you create a measurement feature that references two objects in the subassembly with the feature residing in the top level assembly. The expression will only update to changes made at the subassembly level.

Position override overview

You can use the override commands to make a component have a different position in a higher-level assembly than in its immediate parent. The new position of the component appears in all parent assemblies of the assembly in which the override is created.

The override commands are also known as position override, component position override, variable positioning, or variable component positioning.

The **Position** column in the **Assembly Navigator** shows you whether the selected component has been overridden, whether the override is implicit (made by NX) or explicit (made by you), and the constraint status of the component.

You can create or save overrides even without write access to lower-level assemblies.

Where do I find it?

Application	Assemblies
Prerequisite	<p>Set Preferences® Assemblies® Interaction to Assembly Constraints.</p> <p>You must select a component that has at least two higher levels of parents. You can select a subassembly instead of a component if you do not select any of the subassembly's children.</p>
Shortcut menu	Assembly Navigator® right-click a component node® Override Position

Uses for position overrides

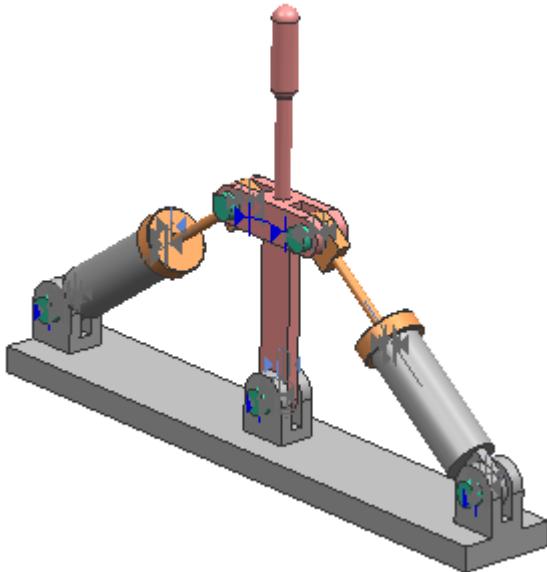
Overrides are useful whenever you have a component that has a different configuration in its parent assemblies than it has when considered in isolation. Some examples are:

- A clamp subassembly. When this subassembly is added to an assembly, one of its components must be overridden to a position where the clamp can grip the assembly geometry correctly.
- A piston engine case. The subassembly containing the piston and conrod is constrained four separate times to a crankshaft component, and each of the four instances requires different positions for the pistons and crankshafts.
- Any subassembly that is constructed in one configuration, but whose configuration is altered when it is installed in an assembly.
- Modelling any subassembly that must be articulated or repositioned as part of its normal functioning. For example, a boom subassembly may need to be modelled in both extended and retracted positions.

Activities: Assembly Arrangements

In the *Assembly Arrangements* section, do the activity:

- *Assembly arrangements with positional override*



Summary

Arrangements provide powerful tools to design and illustrate mechanisms that move by repositioning components either with basic positioning or within degrees of freedom.

In this lesson you:

- Created and displayed several arrangements.
- Used arrangement properties to ignore assembly constraints and properly reposition a variation of an assembly.

Lesson

22 *Interpart geometry*

22

Purpose

One method of assembly modeling is to build the component part files in context of the assembly.

Objectives

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

- Build associativity across parts in the assembly using the WAVE Geometry Linker
- Edit Linked Geometry
- Edit the Timestamp for a link

WAVE overview

Use the WAVE commands to:

- Link geometry between any two part files.
- Get information about the linked geometry and parts.

Links are typically associative; however, there are options to create non-associative links.

After you have copied geometry, you can use the copies in modeling operations even if the part containing the parent defining geometry is not loaded.

Typically, you use an assembly structure to manage relationships between parts containing linked geometry.

You can create a linked part that is *not* a member of the current assembly structure. Using a component member of a WAVE-linked assembly as the seed, you can create one or more linked start parts. Start parts are used to start a new design much like using a template part, but you can update start parts at your discretion and under your full control, any time you change their parent seed part.

Localized interpart modeling

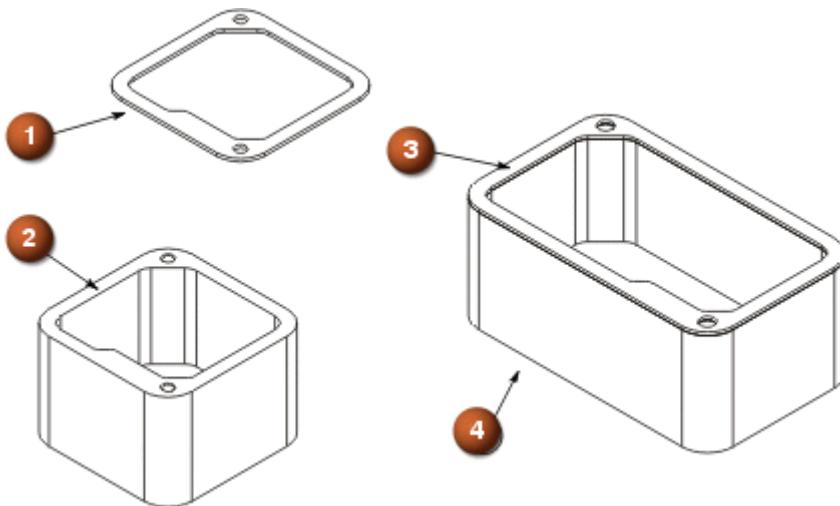
Localized interpart modeling is the ability to relate the geometry of interacting parts in an assembly. This has two distinct advantages in assembly modeling:

- Reduces the cost of design changes.
- Maintains design integrity.

This reduces cost since changes made to a single part can be automatically propagated to other related component parts in the assembly. Design integrity is maintained because the parts will always have correct geometric and positional relationships.



A gasket (1) is derived from a parent face (2) in a housing. If the size or shape of the parent face changes in the housing, the gasket will change accordingly (3) in the assembly (4).

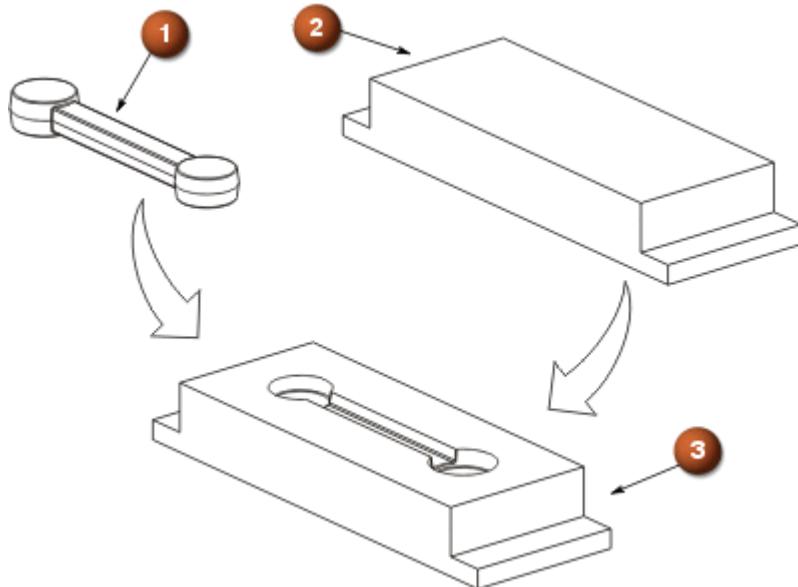


Mold/die applications

Interpart modeling can be applied to create an associative mold or die from a finished piece part.



A solid body (1) can be linked from one part into another (2) where features can be applied to define the cavity in the mold or die (3).





WAVE Geometry Linker overview

Use the **WAVE Geometry Linker** to copy geometry from other parts in the assembly into the work part.

You can create either associative linked objects or non-associative copies. When you edit the source geometry, the associative linked geometry updates.

You can:

- Link geometry from one component part in an assembly into the same assembly work part.
- Link geometry from one subassembly into another subassembly.



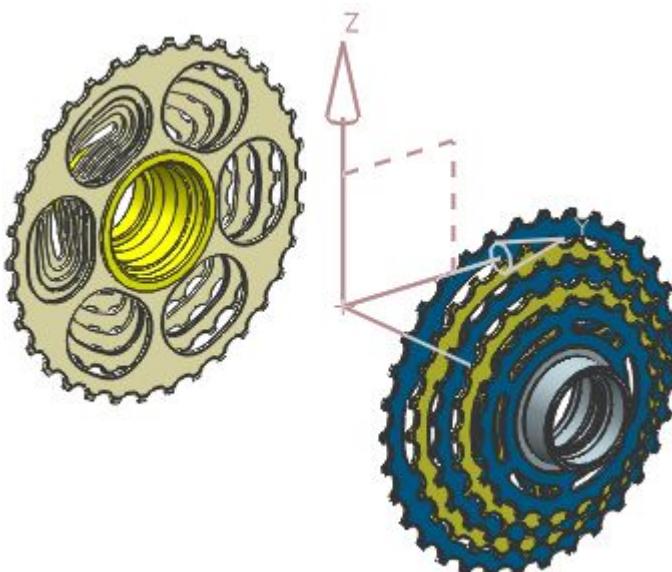
Use the **Tools→Update→Delay Interpart** command to suspend the update of edits to associative links.

To update linked objects, choose **Tools→Update→Update Session**.

All WAVE-linked objects are initially positioned relative to the source geometry and placed on the work layer of the displayed part. Linked bodies are added to the Model reference set in the work part.



The positional and geometric changes made to the associative mirrored bodies on the left updates with the source geometry on the right.



Where do I find it?

Toolbar	Assemblies→WAVE Geometry Linker
Menu	Insert→Associative Copy→WAVE Geometry Linker



WAVE geometry selection

You can create WAVE links with the following types of geometry :

- Composite Curves
- Points
- Datums
- Sketches
- Faces
- Regions of faces
- Bodies
- Mirror bodies
- Routing objects

Routing segments are NX smart curves and include additional connectivity information. Creating a WAVE link for a routing segment creates of two objects:

- The routing segment extracted curve.
- A new segment in the work part constrained to have the same shape as the extracted curve.

Linked objects take the name of the parent object with the prefix **linked** added to the object name. If the object also contains a feature name, that name is added as a suffix to the linked object.

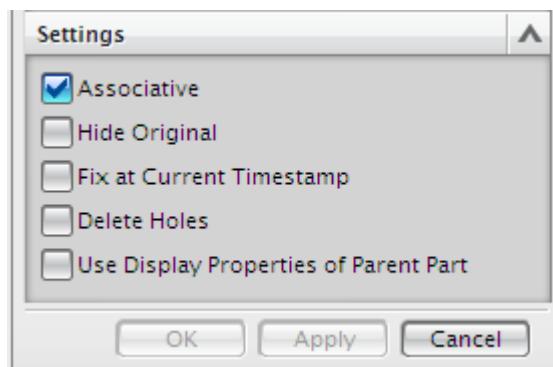


For example, 'Sketch (3) "S21_POCKET" is WAVE-linked into another part file. The new feature displays in the receiving part file **Part Navigator** as 'Linked Sketch (3) "S21_POCKET:S21_POCKET".

WAVE Geometry Linker Setting options

The following common options are found in the **Settings** group of the **WAVE Geometry Linker** dialog box.

- **Associative** makes the linked feature associative to the parent geometry.
Clear the **Associative** check box if you want to copy the geometry into the work part
- **Hide Original** hides the source geometry after the linked geometry is created.
Appears when **Type** is set to **Composite Curve, Datum, Sketch, Face, Region of Faces, or Body**.
- **Fix at Current Timestamp** sets the location of the linked feature in the **Part Navigator** feature creation list.
Appears when **Type** is set to **Composite Curve, Point, Face, Region of Faces, Body, or Mirror Body**.
- **Delete Holes** performs a quality check and removes any holes from the linked faces that result from the operation.
Appears when **Type** is set to **Face or Region of Faces**.
- **Use Display Properties of Parent Part** creates the linked data with the display properties of the parent part instead of inheriting the display properties of the work part.





Wave Geometry Linker general procedure

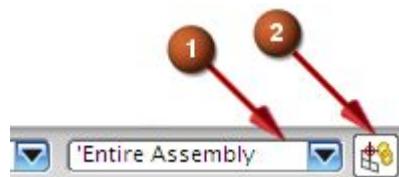
This example shows how to create a WAVE-linked feature with a general procedure.

1. In the **Assembly Navigator**, right-click the component you want to have own the linked geometry and choose **Make Work Part**.
2. On the **Assemblies** toolbar, click **WAVE Geometry Linker** , or choose **Insert→Associative Copy→WAVE Geometry Linker**.
3. On the Selection bar, from the Selection Scope list, select an option based on the component type you want the linked geometry to reside in.
4. In the **WAVE Geometry Linker** dialog box, from the **Type** list, select the type of object that is the source geometry.
5. In the graphics window, from the parent part, select the geometry that you want linked into the work part.

 It is possible with some WAVE Geometry Linker selection steps to have more than one parent part.
Plan how each relationship is developed before you create the link.
6. Select all the options that are relevant to the linked object type and click **OK**.

Design in context WAVE selection scope

The **Selection Scope** (1) and **Create Interpart Link** (2) options on the Selection bar help you design in the context of an assembly.



You can select geometry directly from other components and automatically create WAVE links during many Modeling commands, depending on your **Selection Scope** and **Create Interpart Link** settings. This creates an associative relationship between parts, so that changes you make to one part can drive the design of another part.

When used in various combinations the **Selection Scope** and **Create Interpart Link** settings have the following effects on WAVE linking:

- **Selection Scope Entire Assembly** with **Create Interpart Link** turned on creates associative WAVE links when you select geometry outside the work part.
- **Selection Scope Entire Assembly** with **Create Interpart Link** turned off creates non-associative WAVE links when you select geometry outside the work part.
- **Selection Scope Within Work Part and Components** from the list, the behavior is the same as **Entire Assembly**, but you are limited to selecting interpart geometry only from children of the work part (where the work part is an assembly).
- **Selection Scope Within Work Part Only** allows geometry selection only from the work part during feature creation.

When you select interpart geometry, Selection Intent rules are supported for the following options:

- **Stop at Intersection**
- **Follow Fillet**
- **Chain Within Feature**



Create Interpart Link overview

Use the **Create Interpart Link** command to create WAVE-linked objects while you define features.

The geometry you select to define the feature is copied as linked geometry.

Some of the feature creation commands that include the **Create Interpart Link** option:

-  **Sketch**
-  **Extrude**
-  **Revolve**
-  **Variational Sweep**

Use the Selection Scope options to filter the selection range for the parent part.

- **Entire Assembly.**
- **Within Work Part and Components.** Use this with subassemblies and its components.
- **Within Work Part Only.**

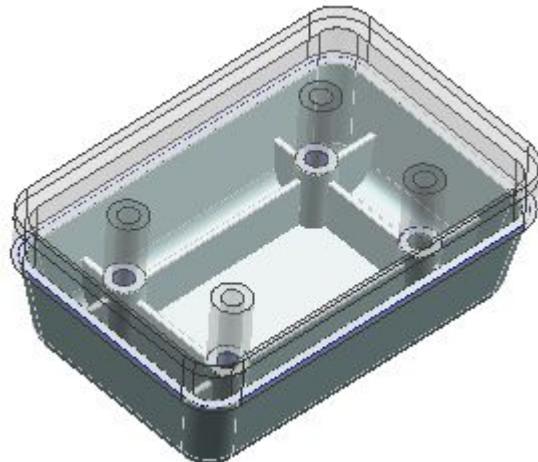
Where do I find it?

Toolbar	Selection bar→Create Interpart Link 
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Activities: Design in context of an assembly

In the *Interpart geometry* section, do the activities:

- *Automatic creation of WAVE links*





Edit WAVE geometry links overview

When you edit most features, the feature creation dialog box appears.

Additional options are available when you edit a WAVE-linked feature:

- Add, remove, or replace linked geometry.
- Edit source geometry from the parent part.
- Map a linked feature to new source geometry.
- Convert a position-dependent feature to a position-independent linked object (PILO).
- Change the timestamp of a linked feature to control the appearance.
- Select a new parent part.

Where do I find it?

Toolbar	Edit with Rollback A small blue square icon containing a yellow wrench and a blue gear.
Menu	Edit→Feature→Edit with Rollback
Graphics window	Right-click→Edit with Rollback



WAVE broken links

After you associatively copy geometry across parts, you can reference the linked data with modeling operations even when the part containing the defining geometry is not loaded.

Occasionally links are broken, usually for one of the following reasons:

- The link is deliberately broken.
- The source geometry is deleted.
- The location of the parent part has been moved or broken. For example, you replaced the component in the assembly structure.

If you perform an edit to the source geometry that could affect linked geometry, a notification box provides additional information.

To prevent the unintentional deletion of source geometry or a parent part, an **Alerts** window in the graphics region provides details of any affected geometry and part files.

You can also identify out-of-date or broken links in the owning component's **Part Navigator**.

- An out-of-date link displays with a yellow bang (!) symbol.
- A broken link displays a single link broken in half.

Use the following methods to repair broken links:

- Redefine the source geometry.
- Re-parent the source geometry or the parent part.
- Remove the linked feature.

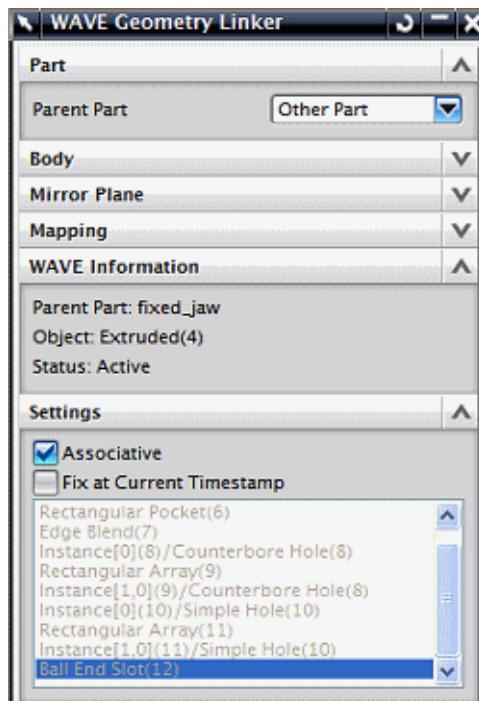
WAVE Geometry Linker edit options

The following is an overview of common WAVE editing options.

- **Part** lets you select a new parent part and source geometry.
- **Mapping** opens the **Replacement Assistant** dialog box to help you edit the source of the linked or extracted feature.
- **WAVE Information** shows the name of the part where the parent geometry is located, the geometry type, and the link status.
- **Settings**

Fix at Current Timestamp clear the **Fix at Current Timestamp** check box to remove the timestamp feature update constraint.

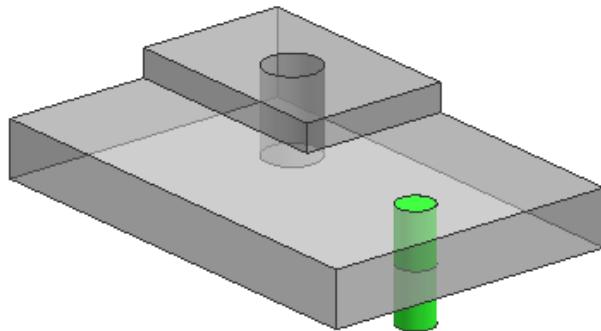
Make Position Independent select this check box to position the owning component of the linked object to be independent from the position of the source geometry. Once you convert to position-independent you can not change it back to position dependent.



Activities: Edit links

In the *Interpart geometry* section, do the activities:

- *Edit a linked composite curve*



Summary: Interpart geometry

Interpart modeling methods allow you to relate geometry in an assembly.

In this lesson you:

- Created components using interpart modeling.
- Built associativity across component parts in an assembly using Geometry Linker.
- Edited the timestamp for a link.
- Edited linked geometry.

Lesson

23 Interpart references

23

Purpose

Interpart References enable components to share parameters.

Objectives

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

- Create and apply referencing interpart references.
- Understand and recognize overriding interpart references.

Interpart expressions overview

Interpart expressions create non-geometric interpart references so you can link expressions from one part to another.

You can create two types of interpart expressions:

- Interpart expressions — A type of interpart reference. The expression references the values in another part if the name of that part appears on the right side of the equation. The expression overrides values in another part if the name of that part appears on the left side of the equation.
- Referencing expressions — An expression or geometry element that references another part. Interpart references can include WAVE links, positioning constraints, and promotions.



When you load a component containing an overriding interpart expression which conflicts with an overriding interpart expression already in your session, the newly-loaded overriding expression is deleted. All expressions referencing the deleted expression are converted to constants.

Syntax

The syntax for an interpart expression is as follows:

`part_name::expression`

In the following example, the interpart expression **diameter** is used to override the value of the expression **hole_dia** in the part named **plate**.

Name	Formula
plate::hole_dia	diameter

The syntax for a referencing expression is as follows:

`expression::part_name`

In the following example, the referencing expression **hole_dia** in one part takes its value by referencing the expression **diameter** in the part named **bracket**.

Name	Formula
hole_dia	bracket::diameter+tolerance

Quoting Filenames

File names must be quoted if they contain one or more of the following characters:

' ' (space) = + - * / () : ^ ! < &

For example, to refer to the expression **length** in a part named **top-level**, create the expression as:

"top-level"::len



Not all of the preceding characters are valid in filenames on every operating system; however, if any of them appears in a filename, that filename, and *only* the filename, must be quoted in expressions to be interpreted correctly.

Where do I find it?

Application	Modeling
Prerequisite	At least two part files must be available.
Menu	Tools ® Expression and then click Create Interpart Reference or Edit Interpart References .
Location in dialog box	At the bottom of the Expressions dialog box.

General concepts

Interpart references (IPRs) allow the user to establish relationships between expressions of separate part files. A change to an expression in one part file may change an expression in a different part file, thus altering the geometry of that part.



IPRs may be created between any two part files, not necessarily between components of an assembly.

23

Your system administrator may choose to disable IPRs at the site, group, or user level.



The interpart expression capability is controlled by a customer default. Choose **File**→**Utilities**→**Customer Defaults**→**Assemblies**→**General**, on the **Interpart Modeling** tab, under **Allow Associative Interpart Modeling**.

There are three possible settings:

- **Yes** — You can freely define interpart expressions.
- **Yes, but Not within Commands** — You can freely define interpart expressions. (The restriction in commands is for geometry linking.)
- **No** — All interpart modeling definition capabilities are disabled.

Overriding expressions

Overriding expressions are interpart references that are created in an assembly to override the value of an expression in one of its components.

The interpart link is on the left side of the equal sign in the expression when viewed outside of the **Expressions** dialog box.

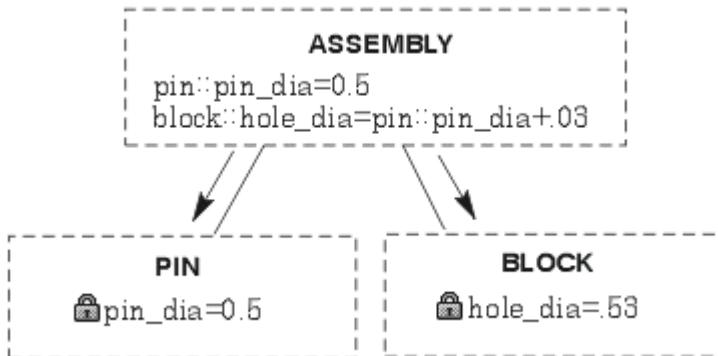
Although they reside in the assembly, they assign a value to an expression in a component part. The expression in the component part will take on this value when the component is opened with the assembly.

The expression being overridden can only be edited from the expression in the assembly which is overriding it.

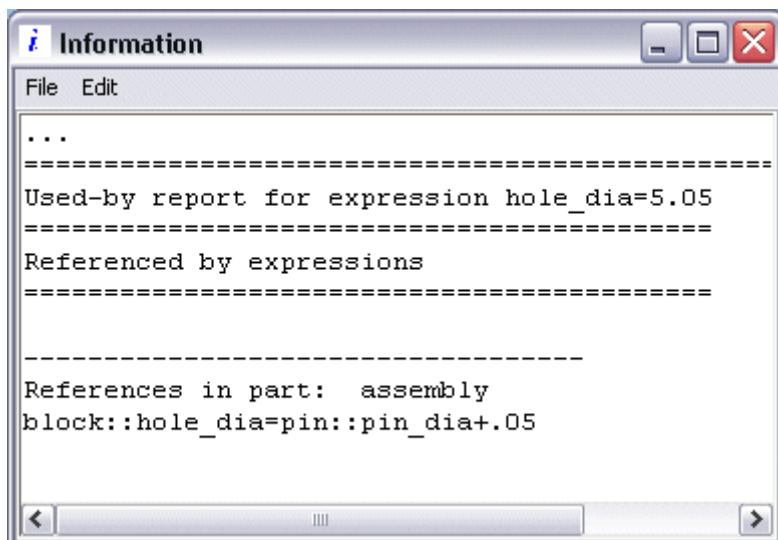


Create overriding expressions in the *Name* box or by selecting the **Use for expression name** check box.

In the example below, the `hole_dia` expression in the block part is being overridden by the expression in the assembly which sets it equal to the pin diameter.



To list references, for a component part choose **Information→Expression→List all by Reference**.



Interpart Update overview

Use the **Interpart Update** commands to delay and update assembly constraints, interpart geometry, and interpart expressions. The **Interpart Update** commands are:

Delay Assembly Constraints

Delays updates to assembly constraints affected by changes you make to your model. To deactivate this command, select it again.

Update Assembly Constraints

Updates all out-of-date constraints in your NX session. This command does not deactivate the **Delay Assembly Constraints** command. If you make additional changes that affect constraints, those constraints become out-of-date again.

Delay Geometry and Expressions

Delays updates to interpart geometry and expressions affected by changes you make to your model. To deactivate this command, select it again.

Update Geometry and Expressions

Updates all out-of-date interpart geometry and expressions in your NX session. This command does not deactivate the **Delay Geometry and Expressions** command. If you make additional changes that affect interpart geometry and expressions, they become out-of-date again.

Update All

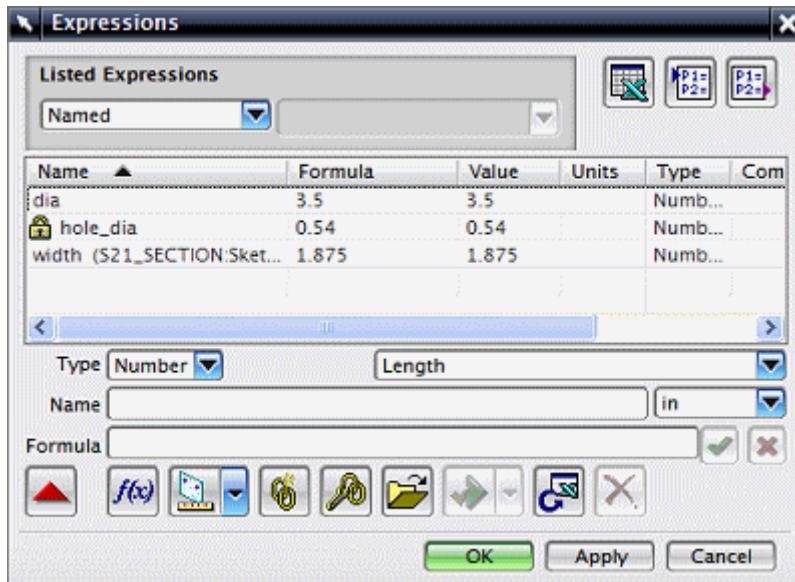
Updates all out-of-date constraints, interpart geometry, and interpart expressions. This command does not deactivate the **Delay Assembly Constraints** and **Delay Geometry and Expressions** commands. If you make additional changes, affected constraints, interpart geometry, and interpart expressions become out-of-date again.

Where do I find it?

Menu	Tools® Update® Interpart Update
------	---------------------------------

Interpart reference options

Interpart references are best created and edited in the **Expressions** dialog box. The lower portion of the dialog box while using “more options” contains interpart references, edit, and open buttons.



Create Interpart Reference



Edit Interpart References

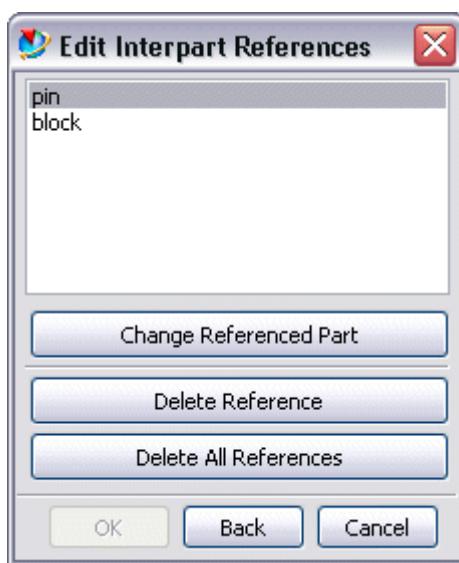


Open Referenced Parts

Edit Interpart References options

You can edit the formula of interpart expressions by the same method you use for any other expression, as described in the Modeling online help. For interpart expressions, you can change the name of a part for all references in your assembly.

In the **Expressions** dialog box, click  **Edit Interpart References** and select the part containing the referenced expression.



Change Referenced Part replaces the selected part name with another part name everywhere in the assembly.

Delete Reference deletes all reference to the selected part name, and replaces all referenced or overridden expression values for the selected part with their current constant value.

Delete All References removes all interpart expressions in the assembly, and replaces all referenced or overridden expression values with their current constant value.

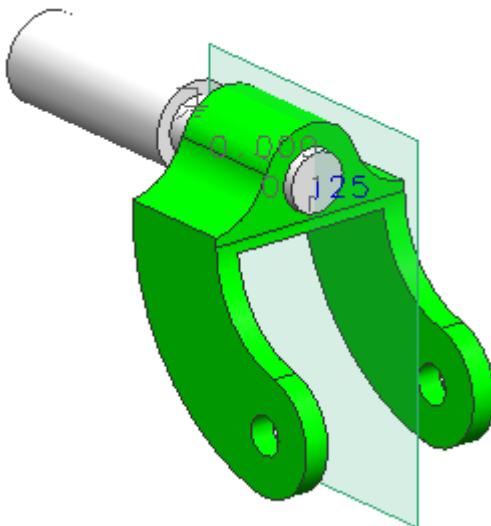


When changing references, the expressions must exist in both parts. If this is not the case, the system will display a message that it will assign the missing expressions their current numeric values.

Activities: Create Interpart References

In the *Interpart References* section, do the activity:

- *Create referencing expressions*



Partial loading issues

Partially loading components in an assembly conserves system memory by not loading all data associated with the file.

When using interpart references, it is possible to edit the expressions referenced by a partially loaded component. The geometry in that part will not update to reflect the changes until the part has been fully loaded.



The **Load Interpart Data** option from **Assembly Load Options** dialog box can be used to ensure all referenced components are fully loaded when partial loading is used.

Resolving interpart expression references

When a part containing an IPR is loaded, the system looks for the name of the expression in the referenced part. If the correct name is found, the system has resolved the link.

If it is impossible for the link to be resolved, the system will notify the user, delete the link, and assign the last known constant value.



Here is an example of an expression in a component part file referencing an expression in an assembly file.

dia=ipr_block_assm::ipr_dia

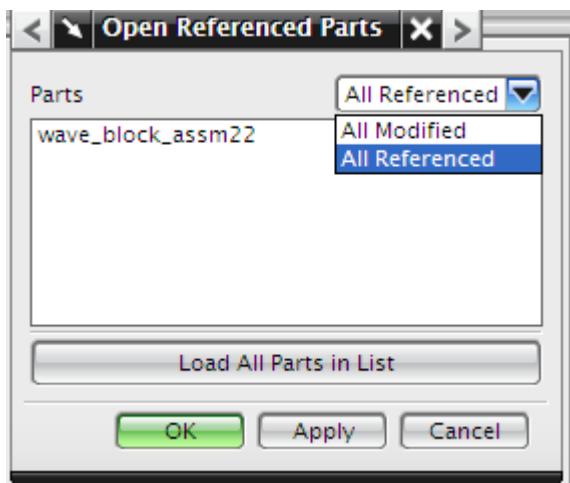
Attempting to delete the expression "ipr_dia" within the assembly part file would result in an error message stating that the expression is in use, and the references would be listed.

If you perform a **Save As** on a part, any loaded part which references it will rename the expression so the link is preserved, now referencing the new part created by the **Save As** operation.

If the other parts are not loaded at the time the Save As was performed, their expressions can be changed later by using **Edit Interpart References** in the **Expressions** dialog box.

Load Parts

When you load a component of a referenced assembly you can list and then open referenced parts from the **Expressions** dialog box with  **Open Referenced Parts**.



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- **Parts→All Modified** lists only those unloaded or partially loaded parts whose expressions have been modified.
- **Parts→All Referenced** lists all unloaded or partially loaded parts with expressions referenced by the work part.
- **Load All Parts in List** fully loads all listed parts.



The **Open→Component Fully** option in the **Assembly Navigator** can also be used to fully load and update interpart references.

Tips and recommended practices

- Before using interpart references, you should evaluate their downstream impacts.
- Do not use IPRs just because you can. They are a very powerful tool that adds another level of complication to the assembly.
- IPRs should be used when the parts have a physical constraint and are used in the same assembly.
 -  Although you can use IPRs with parts not assembled together, it is not recommended.
- Set up company-wide standards on how and when IPR's are to be used.
 -  It may be a good idea to have a naming convention such as a prefix on the expression name such as "ipr_dia" so it is readily identifiable as being referenced.
- Do not use overriding expression references on the same component from different assemblies. This would cause the component to be updated each time it was loaded by the different assemblies.
 -  For this reason, overriding expression should not be used for standard parts such as a bolt or screw.
- In general, it is a good practice to edit IPR's only when all of the referenced parts are fully loaded.
 -  If a part fails to load because of an IPR edit, you should close all other parts then open only the part causing the problem. The part should load and allow you to investigate which expressions are responsible. You can then delete the offending links.

Summary: Interpart references

Interpart references allow you to link the expressions between parts. Whenever a change occurs to an expression in one part file, the related expression in the other part file(s) will change accordingly.

In this lesson you:

- Created and applied interpart references.
- Reviewed tips and recommended practices for using interpart references.

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Lesson

24 Component Arrays

Purpose

Time and effort can be saved by applying component arrays and feature based component arrays.

Objectives

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

- Create a component array.
- Edit a component array.

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Create Component Array overview

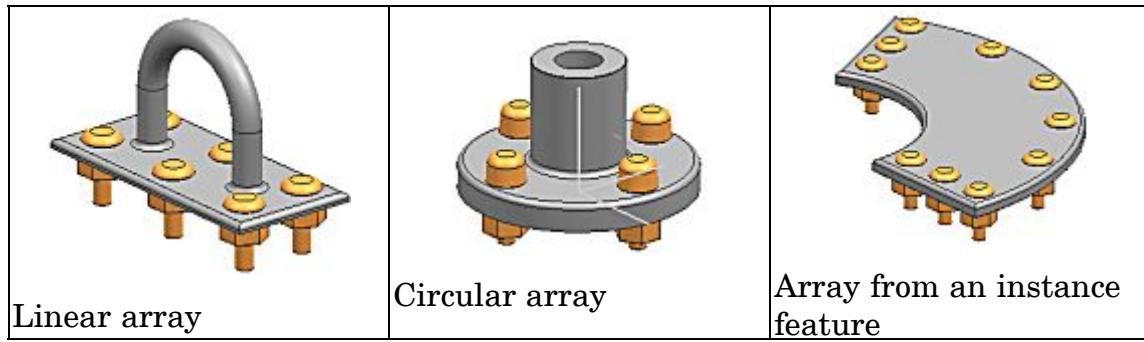
Use the **Create Component Array** command to create named associative arrays of components in an assembly.

Types of component arrays

You can create the following types of component arrays.

- Linear and circular arrays based on a selected master component
- An array that is constrained to instanced feature geometry

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You can use **Create Component Arrays** to:

- Create patterns of components and constraints.
- Add similar components in one step.

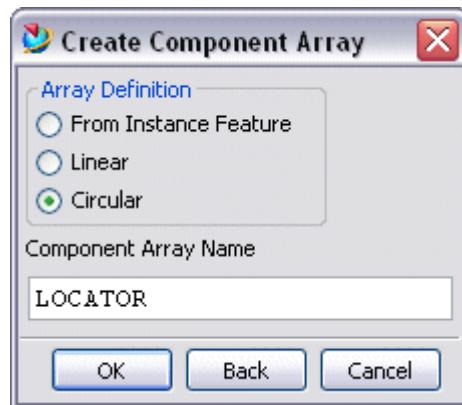
Where do I find it?

Application	Assemblies
Toolbar	Assemblies® Create Component Array
Menu	Assemblies® Components® Create Array

Create Component Array options

Define the type and name of your array with the **Create Component Array** dialog box.

- **From Instance Feature** creates an array based on a template component that is constrained to a feature instance.
- **Linear** creates a linear component array, in either two-dimensional in XC and YC or one-dimensional in XC or YC.
- **Circular** creates a circular array of components from a selected component.
- **Component Array Name** is used to specify a name for the component array. Use up to 30 alpha or numeric characters.



Linear & Circular Arrays

Linear and circular arrays are very similar to feature instancing, except that a linear master component array is not defined by the WCS.

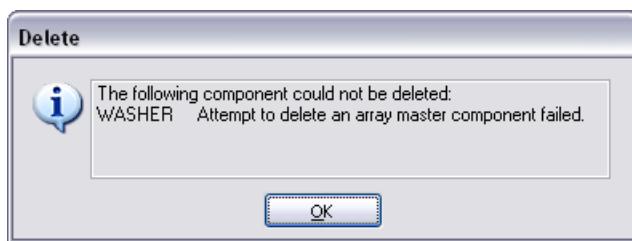
Linear and Circular Array components create:

- New components that are offset from the original component.
- Expressions that control the number of components and the array offsets.
- Associativity to the master component's position to provide updates from possible changes.

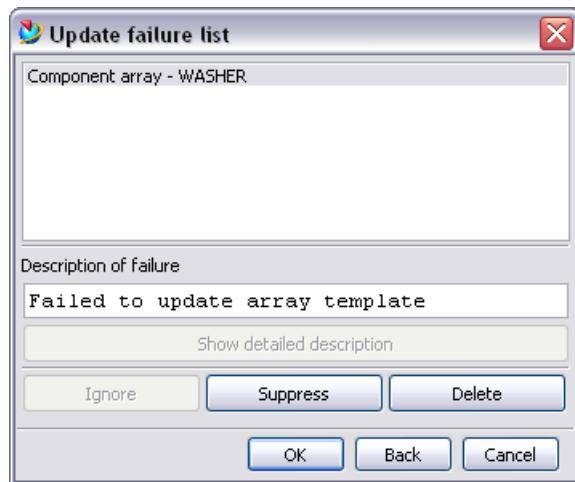


You cannot delete a master component without first deleting the array.

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If you attempt to delete the last remaining component in a **From Instance Feature** array the **Update failure list** dialog box displays. Delete the array from the resulting dialog box.

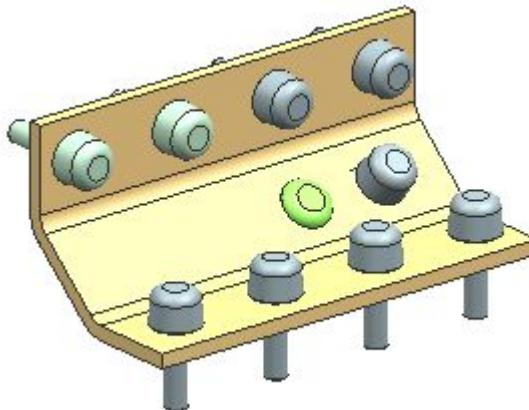


Edit Component Array overview

Use the **Edit Component Arrays** command to modify a component array in the work part.

You can edit components in an array independently.

You can edit a component of an array just as you edit a non-array component. For example, you can change the mating conditions, color, layers, and so on.



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Any new components you create in the same position are based on the template component.

Any modifications to components are lost if the component is deleted.

Where do I find it?

Application	Assemblies
Prerequisite	The assembly must already contain a component array.
Menu	Assemblies→Components→Edit Component Arrays

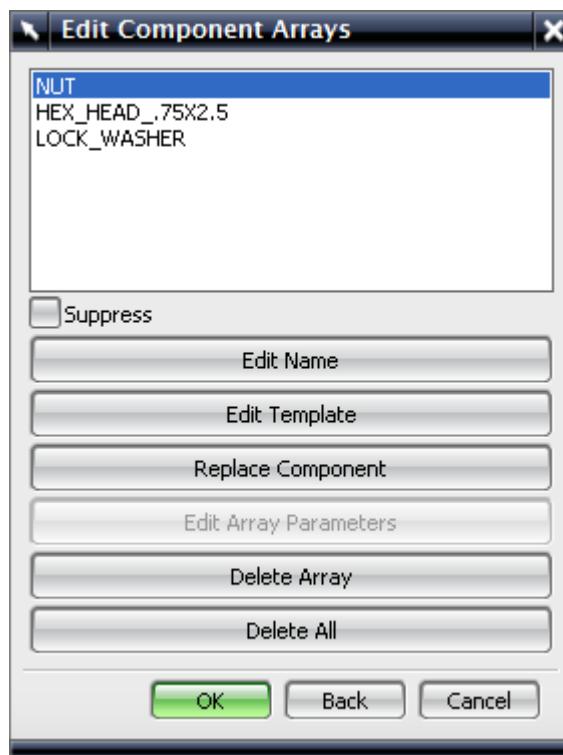
Edit a component array

Use the **Edit Component Arrays** command to:

- Change the number of components.
- Change the array offsets.
- Redefine the direction reference.



The number of components and spacing values are stored as expressions. They can also be edited from the expression editor by pressing Ctrl+E..

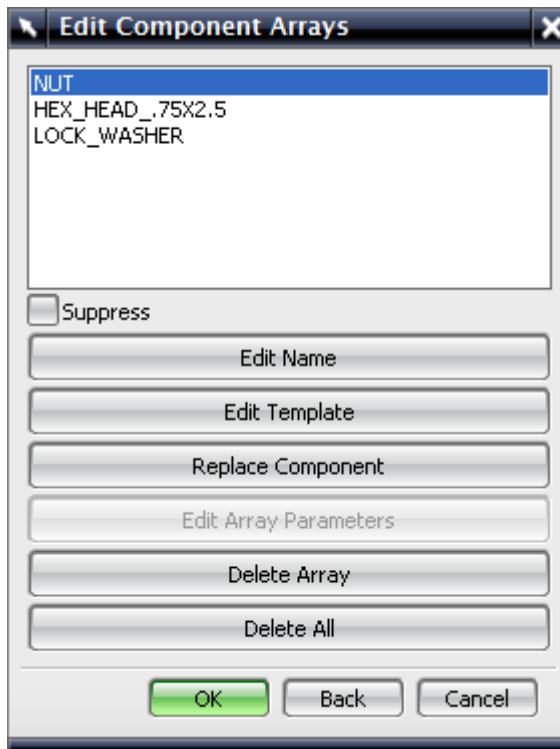


Delete an Array

There are two options available for deleting a component array.

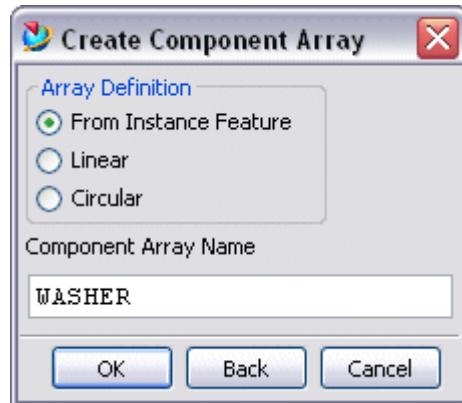
Delete Array Deletes the selected component array parameters, but all of the arrayed components are retained.

Delete All Deletes the selected component array parameters and the array's components. The original template component cannot be deleted unless the component array has been deleted first.



Feature-based component arrays

In many cases it is necessary to associate an array of components to a corresponding array of features in another component of the assembly, for example bolts associated to a hole pattern. This can be accomplished with the **From Instance Feature** option in the **Create Component Array** dialog box.



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The template component

Component arrays produce occurrences of a "template" component object. These occurrences are all associated to the template component. Any changes made to the original component are reflected in the occurrences of the component.

The template component defines certain properties for any newly generated occurrences within the array which include:

- component part
- color
- layer
- name

Component Arrays and Assembly Constraints

The **From Instance Feature** option will generate assembly constraints for the new occurrences based on those of the template component.

When working with the assembly constraints for From Instance Feature arrays, you will:

- Apply at least one assembly constraint to an object belonging to an instanced feature.
 -  This constraint is how NX knows to use the Instance Array to define the Component Array.
- Create the assembly constraints to the template component *before* creating the array.
- Define the assembly constraints to the original feature that was instanced in the component part, if possible.

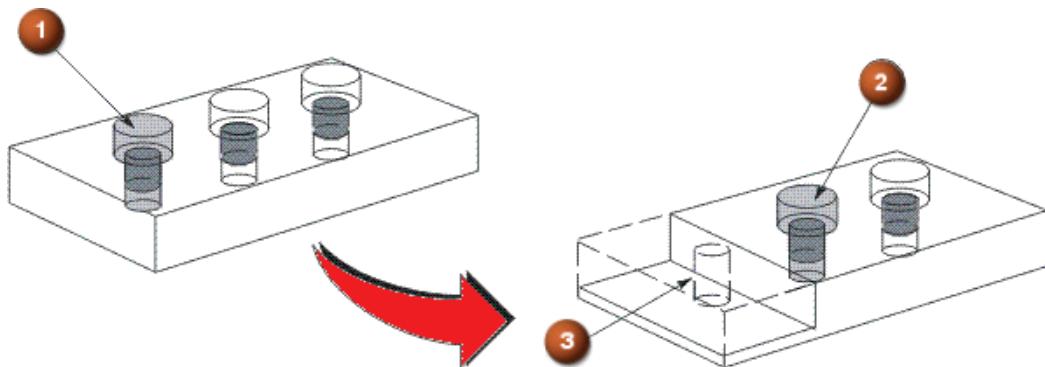
Feature-based array associativity

If the number of features in an instance set is changed, the components in the array associated to those features also changes.

If a feature in an instance set is removed entirely as a result of a modeling change, the corresponding component in the array is also be removed.



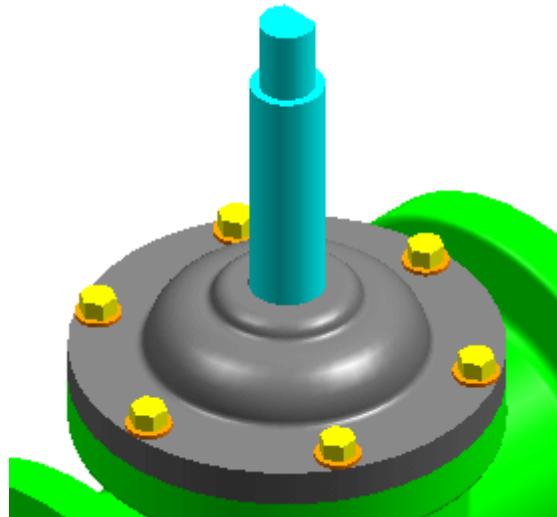
A modeling change causes a hole to be removed (3). If the deleted component was the "template" (1), the system assigns a new template (2) from the remaining components in the array.



Activities: Create component arrays from feature instances

In the *Component Arrays* section, do the activity:

- *Create component arrays from feature instances*



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Summary: Component Arrays

Component arrays take advantage of existing parametric data and can save time in adding component part files to an assembly.

In this lesson you:

- Created a component array.
- Edited a component array.

Lesson

25 Reuse Library

Purpose

The Reuse Library provides methods to quickly define similar parts based on a single template part. In this lesson, you will learn how to define and add a reused sketch.

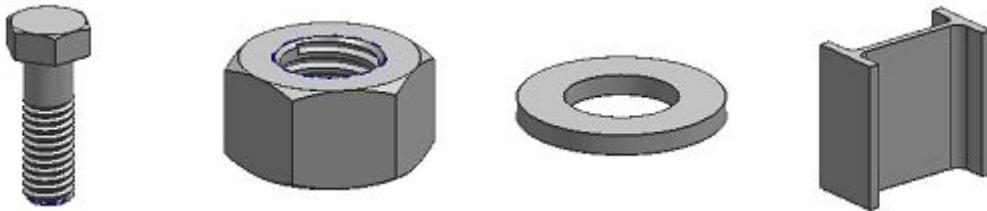
Objectives

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

- Create a 2D sketch template for reuse.

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Reuse Library overview



Use the **Reuse Library** navigator to access reusable objects and to insert them to your model. The objects include the following:

- Industry standard parts and part families
- NX machinery part families
- Product Template Studio template parts
- Routing components
- User defined features
- Law curves, shapes and profiles
- 2D sections
- Drafting custom symbols

The Reuse Library also supports knowledge enabled part families and templates. When you add a reusable object to your model, the dialog box that opens depends on the type of the object. For example, if you add a knowledge-enabled part or part family from the **Reuse Library** navigator, the **Add Reusable Component** dialog box opens.

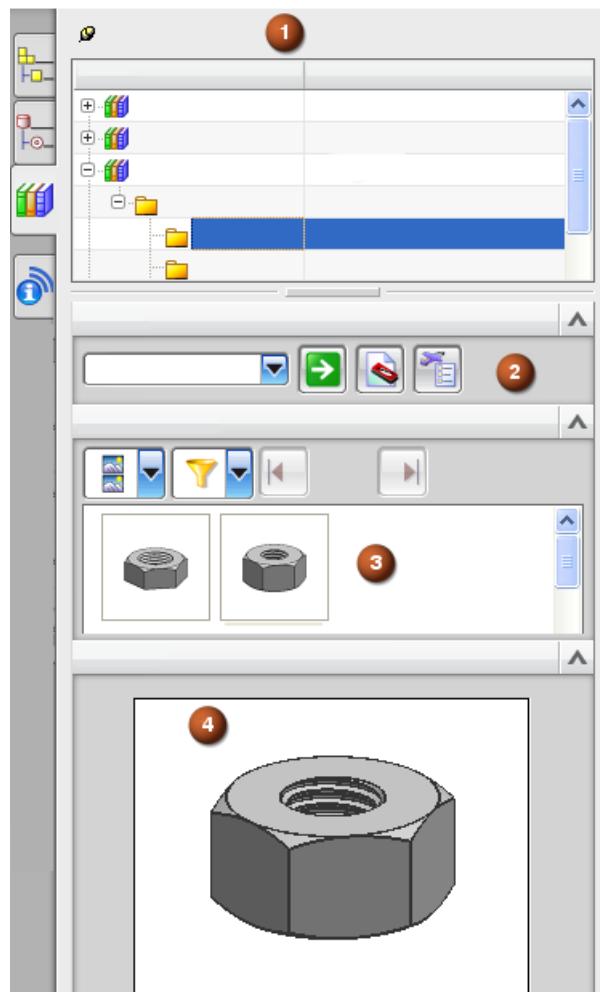
You can use the Reuse Library in native NX , or in Teamcenter Integration mode. In Teamcenter Integration, classification groups are also available from the Reuse Library navigator.



Reuse Library navigator overview

The **Reuse Library** navigator is a NX resource tool like the **Assembly Navigator** or **Part Navigator** that displays reusable objects in a hierachal tree structure.

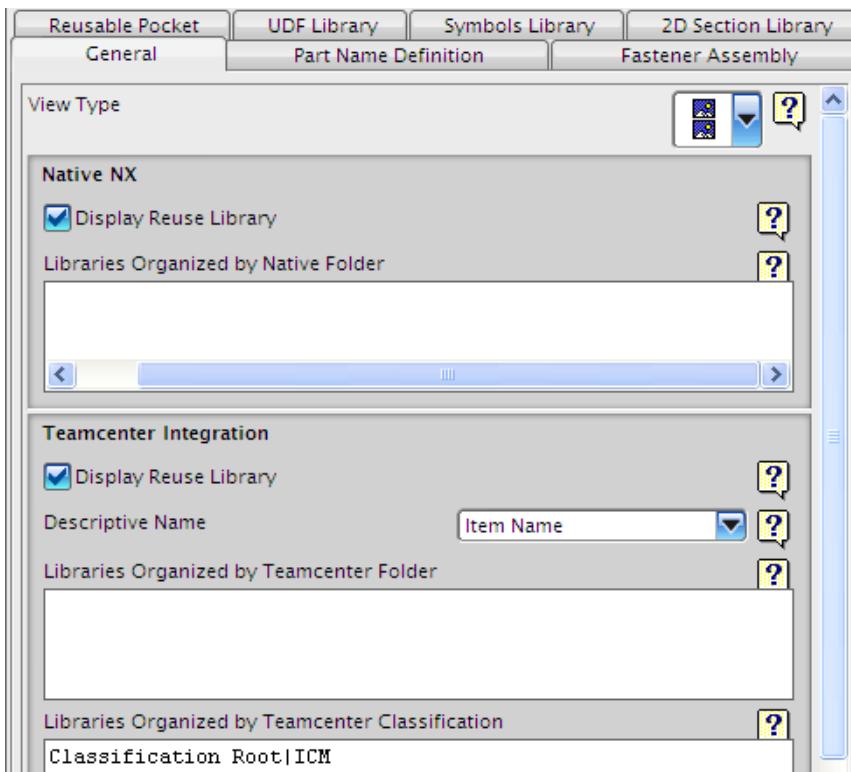
1. Main panel displays the library containers, folders and subfolders they contain.
2. **Search** panel lets you search for objects, folders, and library containers.
3. **Member Select** panel displays the objects and subfolders in the selected folder and displays the search results when you perform a search.
4. **Preview** panel displays the saved preview for the object selected in the **Member Select** panel.



Display the Reuse Library

To display the library containers in native NX or Teamcenter Integration, you must set the directory path in the **Customer Defaults** dialog box.

1. Choose **File® Utilities® Customer Defaults**.
2. In the **Customer Defaults** dialog box, click **Gateway® Reuse Library**.
3. On the **General** page, in the **Native** and **Teamcenter** groups, select the **Display Reuse Navigator** check box.
4. Click the **Part Name Definition** tab.
5. In the **Native NX** group, specify the **Primary Save Directory** by typing a path into the box, or click **Browse** to search through your OS folders.

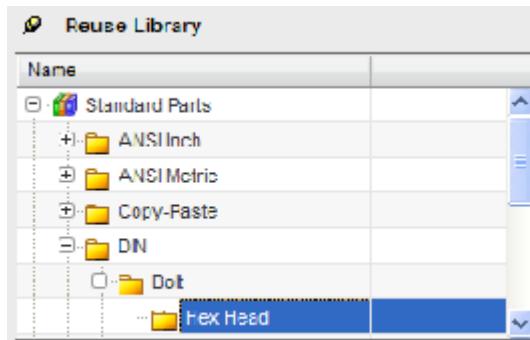


If you need help on any of the defaults, put your cursor over the Help icon .

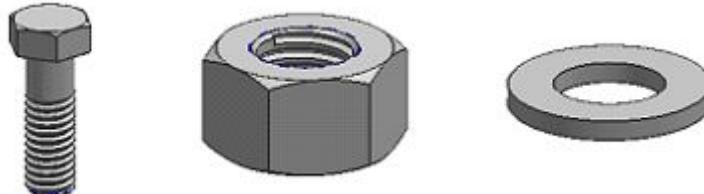
Machinery Library overview

What is it?

The **NX Machinery Library** includes an extensive set of industry standard parts. These high quality, up-to-date models support all major standards – ANSI inch, ANSI metric, DIN, UNI, JIS, GB, GOST and are designed to integrate with the **Reuse Library**.



All parts in the **NX Machinery Library** are Knowledge Enabled parts and integrate with the **Add Reusable Part** dialog box for smart insertion of parts into assemblies.



Why should I use it?

Use the industry standard parts in the **NX Machinery Library** rather than designing your own. You can save even more time if you use the **NX Machinery Library** in conjunction with the **Reuse Library** when you add parts into your assemblies.

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Where do I find it?

The **NX Machinery Library** is available to any customer with an NX Mach license under maintenance. Download the library through the Full Product Download section of the Customer FTP site.

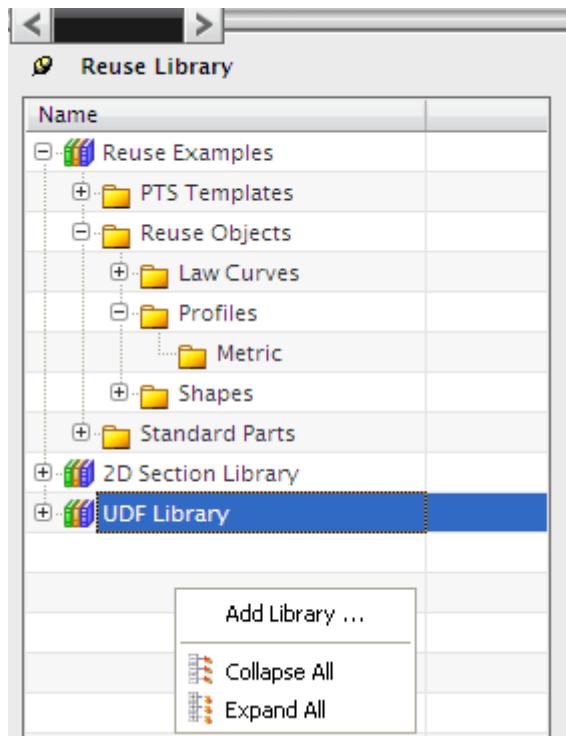


You are not obligated to download or use the **NX Machinery Library**.

Add a library container to the Reuse Library

If the **Reuse Library** navigator is empty, you need to add a library container before you can find and use parts.

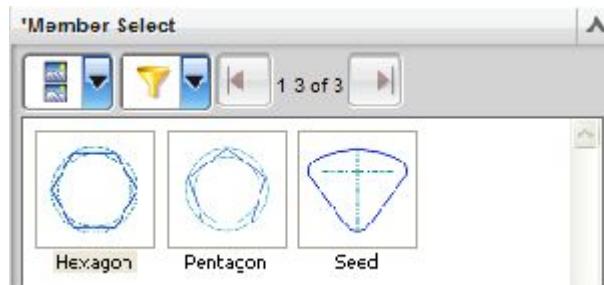
1. Right-click in the background of the main panel and choose **Add Library**.



2. In the **Choose Directory** dialog box, navigate to the directory that contains the parts you need.
3. Click **OK**.

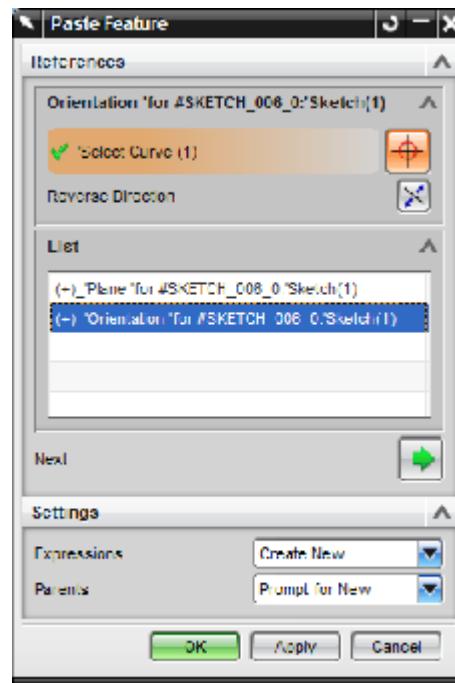
Add a reusable object to a model

1. Navigate to the library container that contains the reusable object.
2. Drag the reusable object from the **Member Select** panel to the graphics window.



You can also right-click the object and choose **Add to Assembly** or **Insert**.

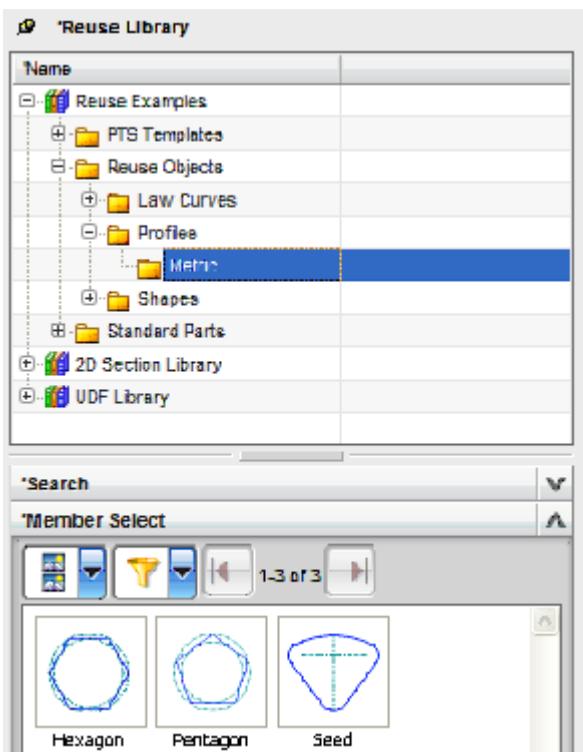
3. Use the appropriate dialog box to position the reusable object.
- For faces and bodies, if you drag the object to a face or a datum, NX selects the cursor location as the anchor point for the object and aligns it with the normal vector of the highlighted face or the datum.
4. Optional: In the **Paste** dialog box, select **Orientation** and define the vector reference.





Define Reusable Object overview

Use the **Define Reusable Object** command to save a frequently used feature or object from your model as a reusable object template in the Reuse Library.



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Such objects include the following:

- Faces
- Bodies
- Features
- 2D sections
- Additional types specified by the global selection filter

Where do I find it?

Toolbar	Reuse Library ® Define Reusable Object 
Menu	Tools ® Reuse Library ® Define Reusable Object

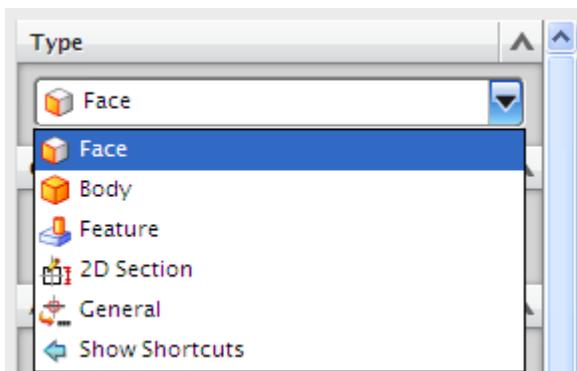


Define a reusable object

1. Identify the object you want to define as a reusable object and make the part that contains it the work part.

2. On the **Reuse Library** toolbar, click **Define Reusable Object** , or choose **Tools ® Reuse Library ® Define Reusable Object**.

3. In the **Type** group, select the type of object you want to define.



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4. In the **Object** group, click **Select Object** , and select one or more objects from the work part.

5. In the **Anchor** group, click **Specify Orientation** .

6. In the graphics window, select an anchor point for the reusable object.

7. In the **Folder View** group, select the folder to which you want to save the reusable object.

8. In the **Name** group, type a descriptive name and file name for the reusable object in the appropriate boxes.

9. In the **Preview Image** group, from the **Input** list, select **Graphics Area**.

10. Click **Define Image** .

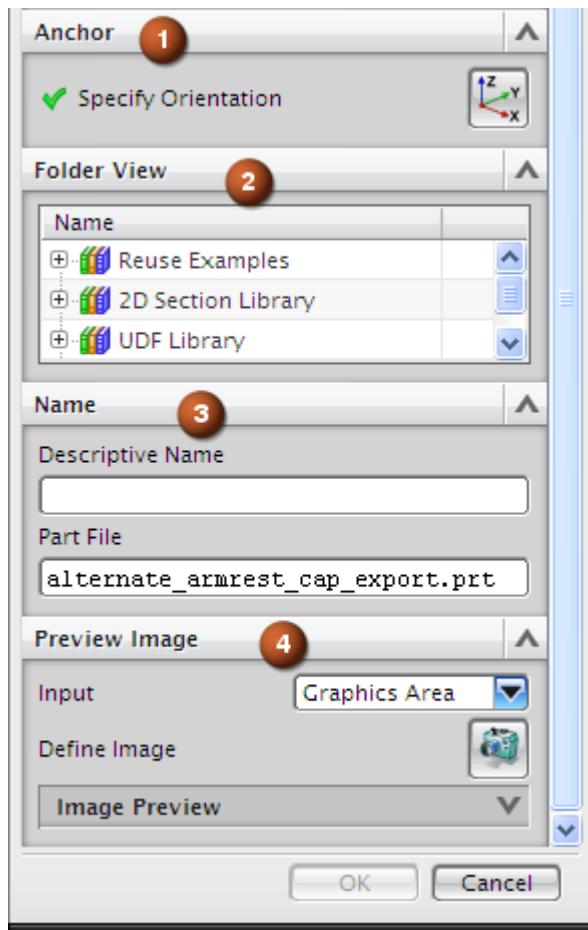
A preview image of the reusable component displays in the preview panel.

11. Click **OK**.

Define reusable object options

The following is a list of options that are unique to defining a reusable object.

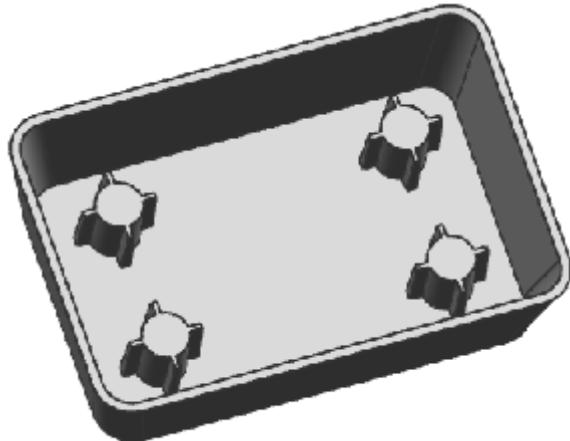
1. **Anchor** specifies fan anchor point for the reusable object.
2. **Folder View** lists the folders you can select from to save the reusable object.
3. **Descriptive Name** and **Part File** lets you define the descriptive and part file names for the object.
4. **Preview Image** contains methods you can use to create the preview image for the reusable object.



Activities: Define and add a reusable 2D section

In the *Reuse Library* section, do the activity:

- *Define and add a reusable 2D section*



Summary: Reuse Library

The Reuse Library provides a method to quickly define similar parts based on a single template.

In this lesson you:

- Created a 2D sketch template for reuse.

Lesson

26 Revise and replace components

Purpose

After creating an assembly, you may have to revise or replace an existing component or change the name of the component part. In this lesson you will investigate the different methods to revise components and the assemblies that use them.

Objectives

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

- Revise a component and an assembly using Save As.
- Replace components in an existing assembly.
- Use various assembly reports.
- Close and reopen part files.

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File Versioning/Revisions

Track revisions by part number

The most common method to track revisions to a component after it has been released is to reflect the revision in the part name.

When revising, the user would save the part with the same base name, but modify the revision identifier. This method is very efficient because you can easily identify the version of a loaded part from the name.

There are several advantages to this method.

Advantages

- Easy to create the change. Use "Save As" on the affected components.
- No file protection problem because owner performs the "Save As".
- The old and new parts can and should reside in the same directory.
- The legacy information is accessed (if on line) by retrieving the older revision assembly or component part.
- Easy to track revisions on the shop floor when looking at numbered parts.
- File versioning rules can be implemented to enable the system to always get the latest version of the file.

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Disadvantages

- If file versioning rules are enabled, two versions of the component part cannot be open at the same time.
- Associated information could be lost if components are substituted and file versioning is not used.

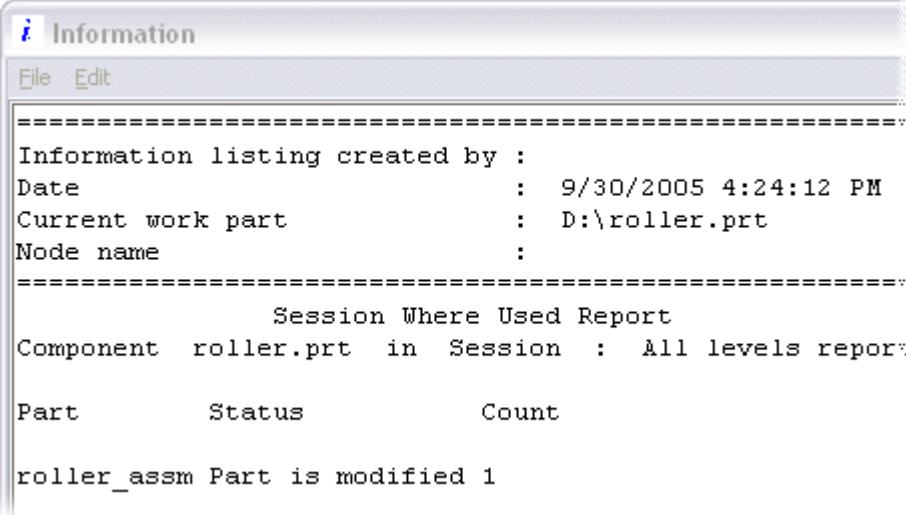
Revise a component and assembly using Save As

When revisions are incorporated into part names, an easy way to revise a component is to save the component with the new name while it is the work part. This can be accomplished by choosing File® Save As.

Many companies, however, require that an assembly also be revised whenever a change is made to the form, fit, or function of one of its components so you would also be required to save each of the assemblies in the tree that reference the component.

When you perform a **File→Save As** on a component part in a native operating system:

- A *Session Where Used* report is immediately displayed, listing any loaded assemblies that reference the component.
- A new name for the component is defined.
- A new name for each of the listed assemblies is defined, as desired.
- An information window is displayed with the new part names.



The screenshot shows a Windows-style dialog box titled "Information". The menu bar has "File" and "Edit". The main area displays session details and a report table. The session details are:

```
=====
Information listing created by :
Date : 9/30/2005 4:24:12 PM
Current work part : D:\roller.prt
Node name :
=====
```

The "Session Where Used Report" section shows:

```
Session Where Used Report
Component roller.prt in Session : All levels report

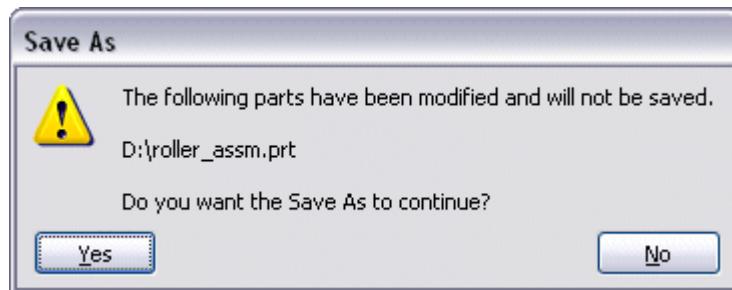
Part Status Count
roller_assm Part is modified 1
```

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Watch the Cue line for a prompt specifying the name of the file currently being renamed.

If you click **Cancel** at any level of the assembly structure, you will get a message after input for the last file has been specified:



Additional Assembly Reports

There are other reporting tools available to help you understand how a particular assembly has changed over time.

- **List Components** — Generates a list of all components in the work part and outputs it to the Information window.
- **Update Report** — This report indicates which components were updated (changed) as the assembly is opened.
 -  An update report can automatically be generated every time an assembly is opened by choosing Preferences® Assemblies and toggling the Display Update Report option to on.
- **Where Used** — This option will search directories and list the assemblies that reference a specified part. This is useful to determine what impact a pending design change may have on other assemblies.
 -  In the Where Used Report dialog box, a Search Option can be chosen to specify what directories to search for parts and whether to list only next level assemblies or all assemblies.



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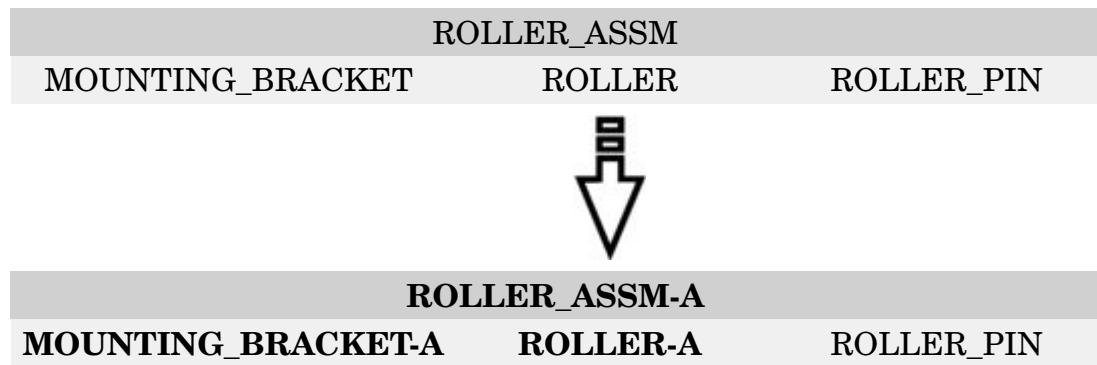
A Where Used report may take considerable time to execute. It is recommended that you search through as few directories and parts as possible.

- **Session Where Used** — This option will list only the loaded assemblies that reference a specified part.
This report is automatically generated when you perform a File® Save As on a component part while the assembly is loaded.

Activities: Revise component using Save As

In the *Revise and replace component* section, do the activity:

- *Revise components using Save As*



Close assembly component parts

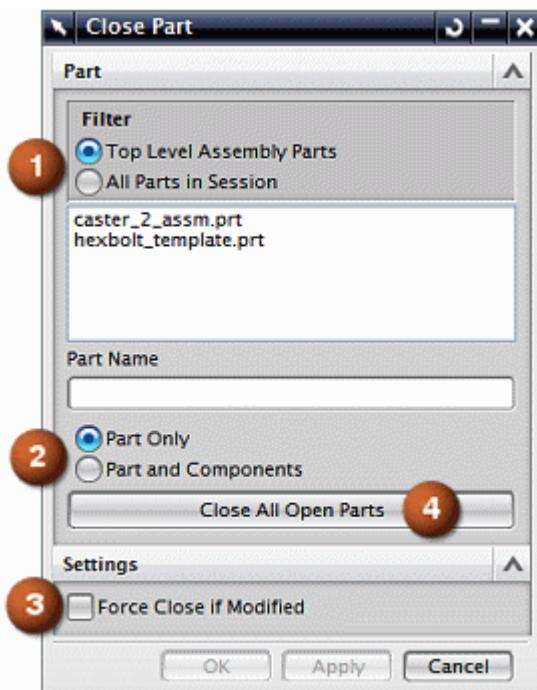
The **File→Close→Selected Parts** option lets you selectively close (unload) components in an assembly.

The upper section of the dialog box lets you specify what component parts to close and the lower section lets you specify how they will be closed.

If the component part you are closing (unloading) has been modified while it was the work part, you will be asked if you really want to close the part.

If you agree to close (unload) the component part, you will lose the modifications you made to the part and the changes won't be reflected in the piece part file stored on disk.

Close Part options



1 – List all loaded component parts or top level only.

2 – Close only parts selected or whole assembly tree.

3 – If on, you will not be warned if a selected part has been modified.

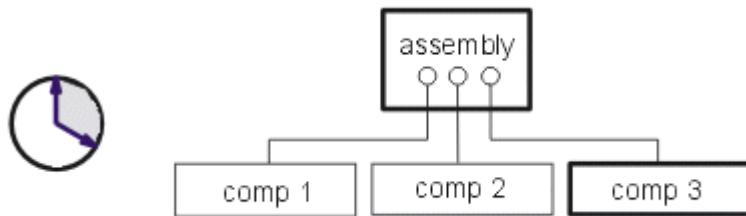
4 – Close all parts in the session.

Reopen component parts

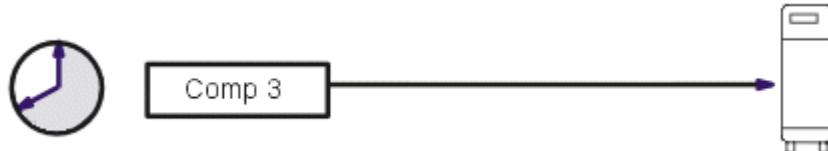
In a concurrent engineering environment, one designer may be working on a loaded assembly which references a component part that another designer is simultaneously modifying.

The **File→Close→Close and Reopen Selected Parts** option selectively updates fully loaded components with their counterparts on disk. It can be utilized in the following situation.

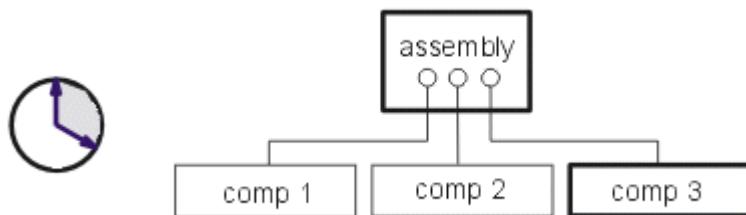
Early in the morning, designer A starts working on an assembly that references comp3.



Later in the morning, while designer A is still working on the assembly, designer B revises comp3 and saves it using File® Save.



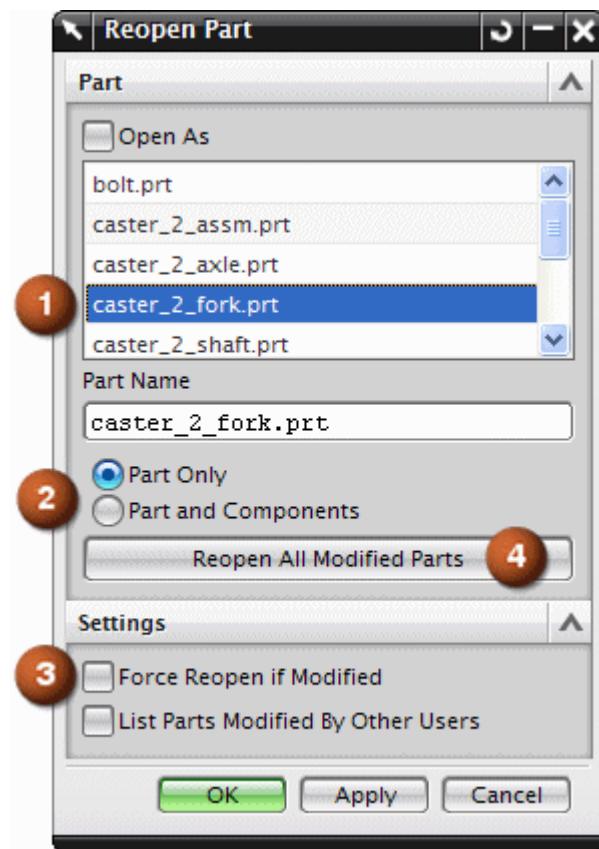
At lunchtime, designer A reopens comp3 while the assembly is still open using File® Close® Reopen Selected Parts.



Reopen Part options

The top portion of the Reopen Part dialog box lets you specify what component will be reopened. The bottom portion of the dialog box lets you specify how the component will be reopened.

After you reopen the parts, an Information window will list the names of the parts, their status before they were reopened, and their status after they were reopened.



1 – List of loaded components that can be reopened.

2 – Specifies whether reopen should affect part or whole assembly.

3 – If on, you will not be warned if selected part has been modified before it is loaded from disk.

4 – Reopens all parts in session that have been changed on disk.



Replace Component overview

Use the **Replace Component** command to remove an existing component and replace it with another component. You have the option to rename the new component.



You must select **Allow Replacement** in the **Assembly Load Options** dialog box, if the new component is:

- Not created from the same template part as the original.
- Not a descendent of the same original blank part if no template was used.

Where do I find it?

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

The Unique Identifier (UID)

When the system finds a component with the correct name, it performs a second check before loading it.

There is an internal file identifier, referred to as a UID (Unique IDentifier), that ensures that the component that has been found is the genuine article, or at least a copy of it.

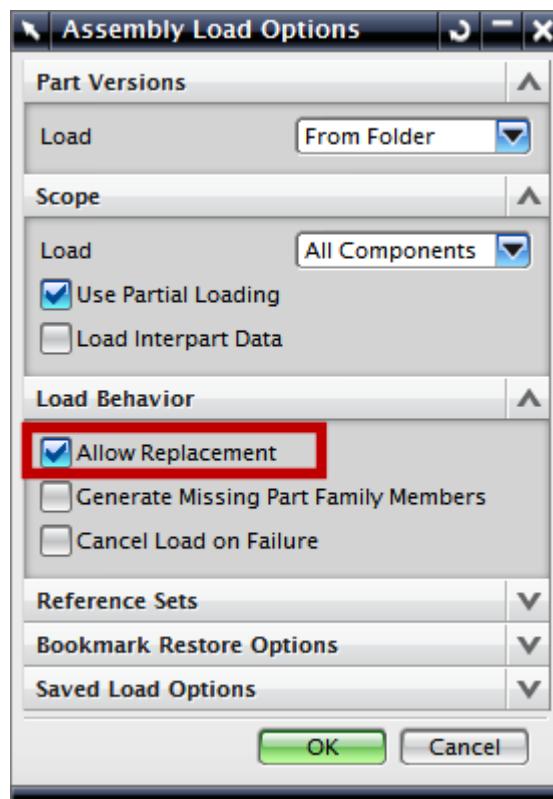
A new UID is not assigned (and thus, associativity is maintained) in the following cases:

- When you copy or move the file in the operating system.
- When you save the file into another directory using the same name.
- When you perform a **File→Save As**, as with a seed part.

Allow Replacement

When you open an assembly and the system finds a component that happens to have the same name but a different UID, the opening will fail unless **Allow Replacement** has been selected in the Assembly Load Options dialog box.

The Allow Replacement option enables a component to be loaded into an assembly even though it has a different UID, or history. It could be a completely different part created by another user.





If the new component has no common history (different UID) with the replaced component, data in the assembly will lose its associativity to the original component (assembly constraints, WAVE interpart references, etc.).

Maintain relationships while replacing a component

In the **Replace Component** dialog box, use the **Maintain Relationships** option to preserve relationships from the original component to the replacement component, to keep as much of the original behavior as possible. The relationships that are maintained vary, based on whether your original and replacement components are different versions of the same part, or different parts.



If the **Maintain Relationships** check box is not selected, the setting of the **Preserve Component Attributes** customer default determines whether or not the component attributes are maintained when a component is replaced.

Your replacement component is a version of the original part file in cases such as the following:

- When you create a new blank part, NX assigns it an internal unique identifier (UID). If, during your design process, you save the part under different names, all the parts have the same UID as the original blank part.
- If you create your part file from a template, all the part files created from that template have the same UID.

When the original and replacement components are different, NX tries to maintain the following relationships:

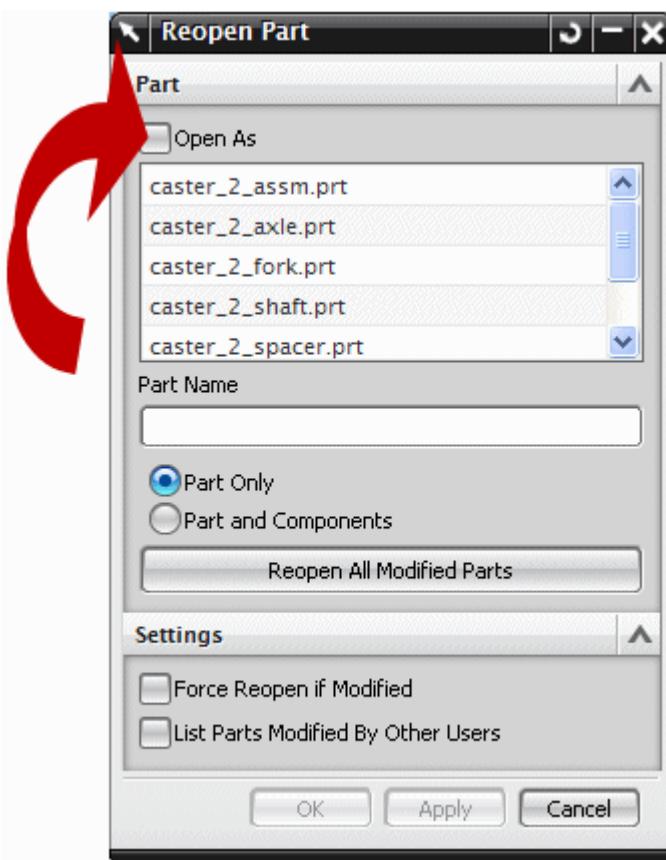
- Transformations
- Color and other component attributes
- Variant condition information
- Occurrence notes
- Variable component positions

When the replacement component is a version of the original part file, NX tries to maintain the following relationships in addition to what has been mentioned above:

- Occurrence information for CAM data, motion data, and so on.
- Assembly constraints
- Mating conditions
- WAVE links
- Interpart expressions
- Drawing views
- Drafting and PMI annotations

Replace components using Reopen

A loaded component can also be replaced with another part by choosing **File→Close→Close and Reopen Selected Parts** and selecting the **Open As** option.



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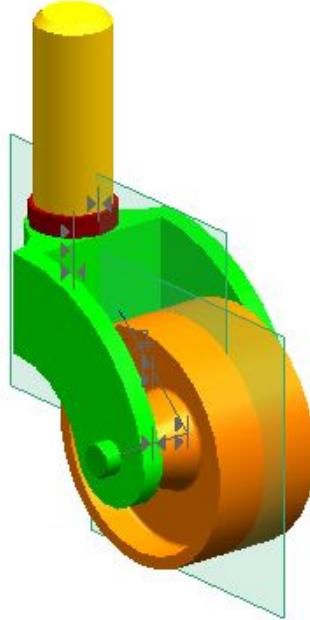
Use the following as guidelines when replacing a component with **Open As**.

- The component you want to replace must be modified in the current session or it will not appear in the **Reopen Part** dialog box **Part** list.
- Once you select the modified component to be replaced you are prompted to select the replacement component.
- If the replacement part has a common history with the original part (same UID), associativity is maintained.
- To reopen a component with a part that has a different UID, **Allow Replacement** must be selected in the **Assembly Load Options**.

Activities: Replace components

In the *Revise and replace components* section, do the activity:

- *Replace components*



Summary: Revise and replace components

After creating an assembly, you may have to revise or replace a component and change the name of the component part. In this lesson, you used different methods to revise components and the assemblies that use them.

In this lesson you:

- Revised a component and an assembly using Save Part As.
- Replaced components in an existing assembly.
- Use various assembly reports.
- Close and reopen part files.

Lesson

27 Introduction to Drafting

Purpose

This lesson introduces the Drafting application and the master model concept.

Objectives

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

- Create a non-master file that references a master model.
- Open, create, and delete drawing sheets.
- Add and edit views on drawing sheets.
- Create dimensions.
- Create notes on a drawing sheet.

Drafting application overview

The Drafting application allows you to produce and maintain industry standard engineering drawings directly from the 3D model or assembly part.

Drawings created in the Drafting application are fully associative to the model. Any changes made to the model are automatically reflected in the drawing.

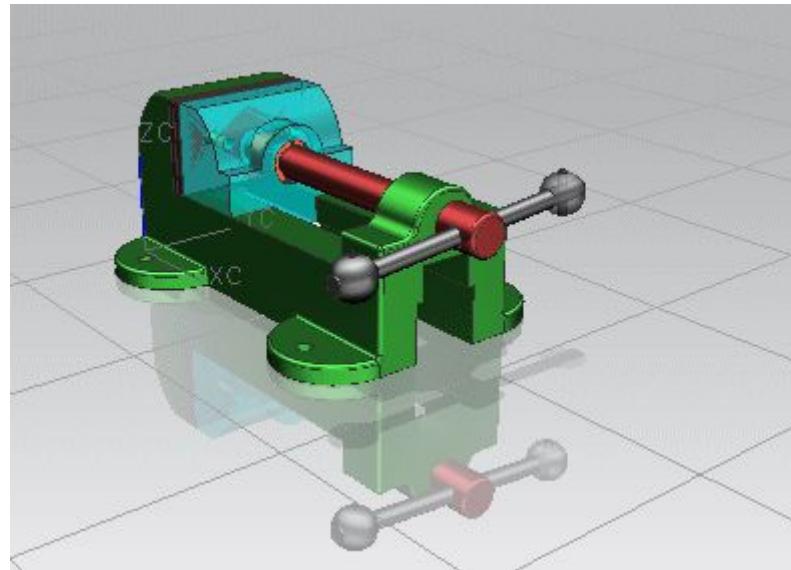
Some of the highlights of the Drafting application include:

- A comprehensive set of view creation tools that support advanced rendering, placement, associative, and update requirements for all view types.
- Fully associative drafting annotation that update when the model updates.
- Controls for drawing updates and large assembly drawings which enhance user productivity.
- Support for major national and international drafting standards, including ANSI/ASME, ISO, DIN and JIS.
- Support for both in-part and concurrent drawing creation in 3D drafting processes.
 - Choose to save the 2D drafting details directly within the part itself, or in a separate part that is fully associated to the master model.
 - The support for concurrent engineering practices enables the drafter to make drawings while the designer concurrently works on the model.

In NX, the term *drawing sheet* is used to define a collection of views. You can think of each drawing sheet as a separate page in the drawing file. One drawing file can contain many drawing sheets.

The 3D drafting process in NX

The following illustrates the general process for creating a drawing from an existing 3D model. This overview is not intended to give a detailed description of specific functions or operations.

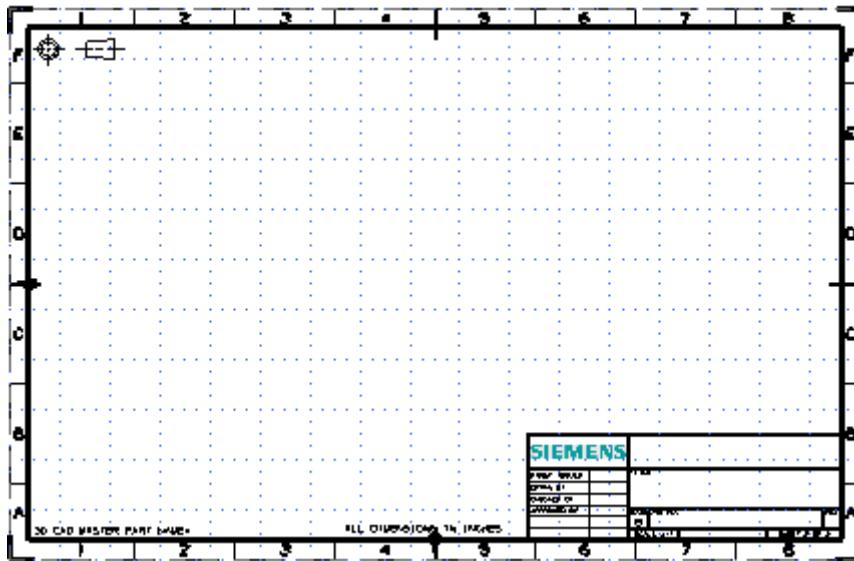


Set your drafting standard and drawing preferences

Before creating a drawing, it is recommended that you set the drafting standard, drafting view preferences, and annotation preferences for the new drawing. Once set, all views and annotations will be consistently created with appropriate visual characteristics and symbology.

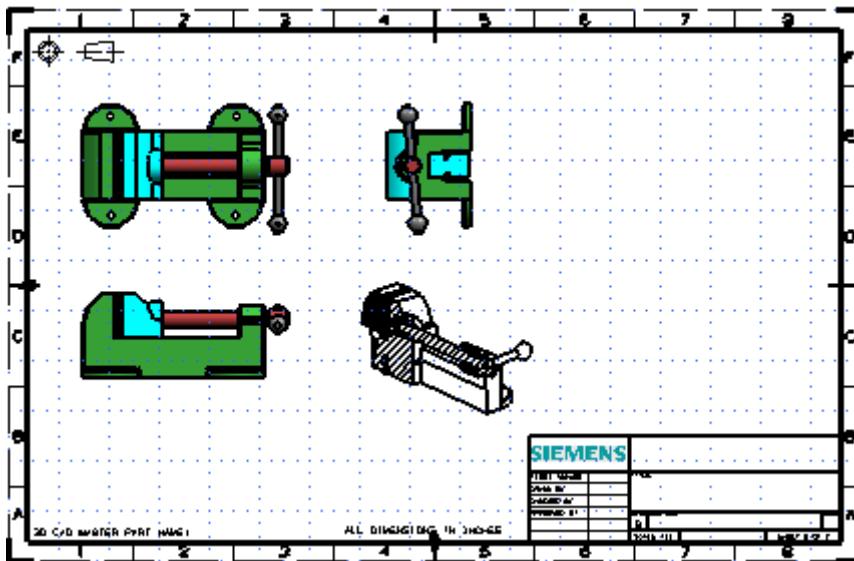
Create a new drawing

The first step in creating a drawing is to make a new drawing sheet either directly within the current work part, or by creating a non-master drawing part that contains the model geometry as a component.



Add views

NX enables you to create a single view or multiple views at the same time. All views are derived directly from your model, and can be used to create other views, such as section and detail views. The base view determines the orthographic space and view alignment for all projected views.

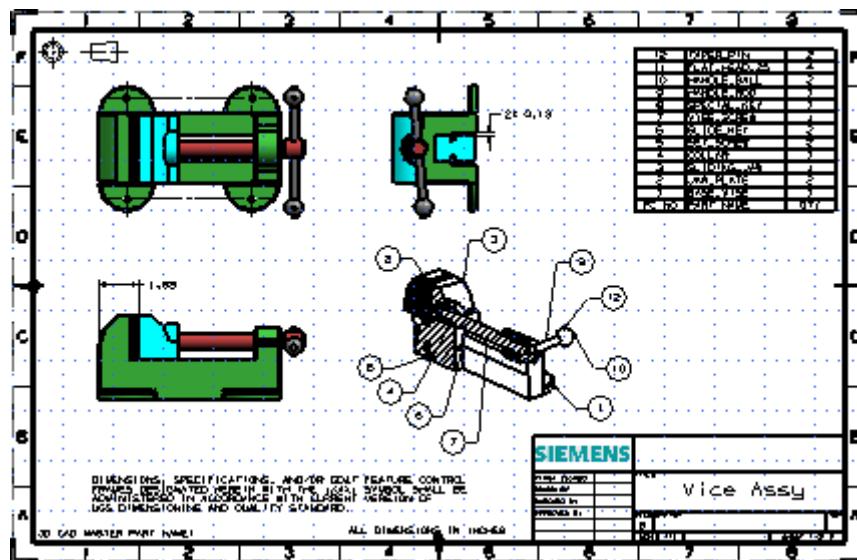


Add annotation

Once you have placed the views on your drawing, you are ready to add annotations.

Annotations such as dimensions and symbols are associated with the geometry in the views. If a view is moved the associated annotations move with the view. If the model is edited, the dimensions and symbols update to reflect the change.

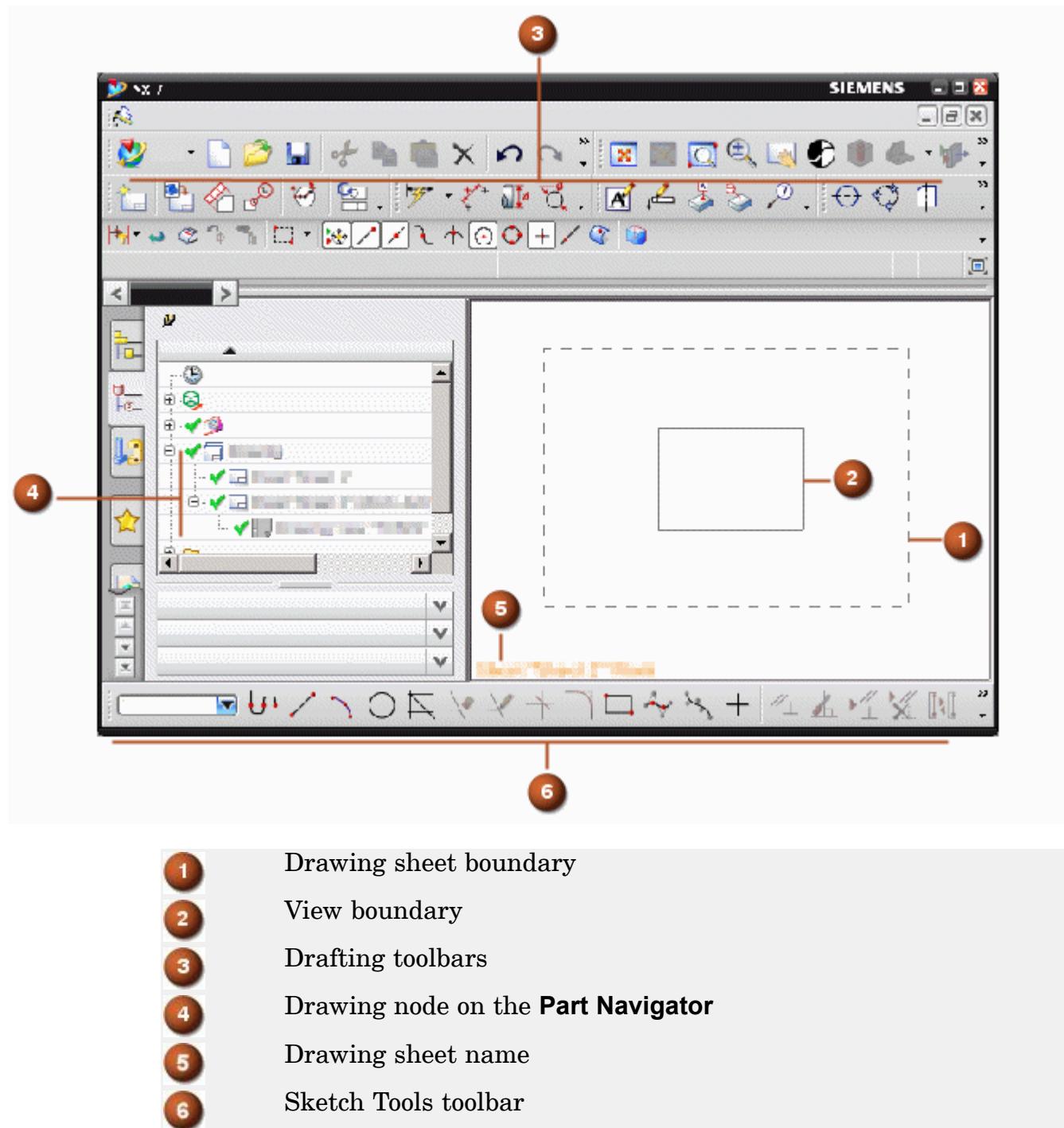
You may also choose to add notes, labels, and in the case of assembly drawings, parts lists to your drawing



A completed drawing can be plotted directly from NX, or the part containing the drawing can be used directly by manufacturing to fabricate the part.

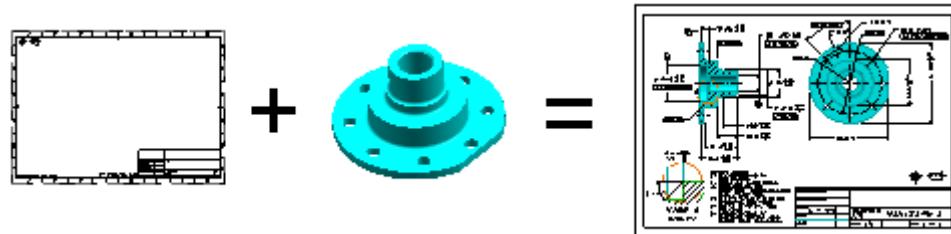
The Drafting interface

In addition to standard tools and selection toolbars, the Drafting user interface contains the following unique features:



Master model concept

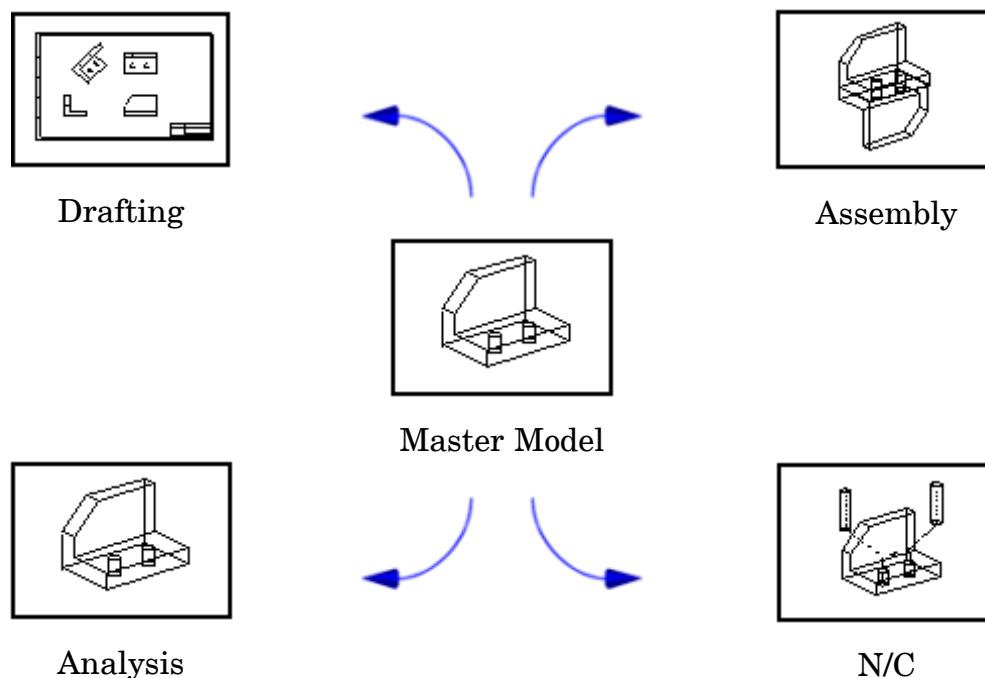
You apply the master model concept by creating an assembly, or non-master part, with exactly one component part. The component part is the master model. Edits to the master model are updated in the non-master part.



Part is added to the drawing file as a component

The master model concept allows multiple design processes to access the same geometry during development. Benefits of this include:

- It promotes concurrent engineering. You can begin downstream applications such as drafting, manufacturing, and analysis during geometry construction.
- The downstream users need not have write access to the geometry. This prevents accidental modifications.

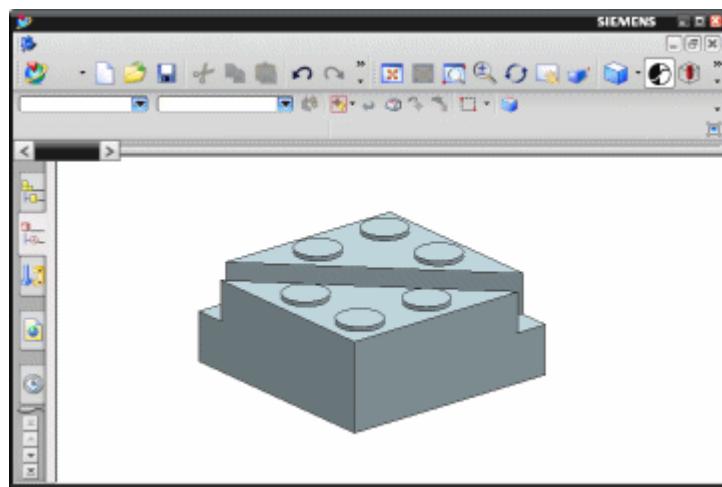


Each application uses a separate assembly part. When the master model is revised, the other applications automatically update with minimal or no associativity loss.

You can maintain the design intent of the various design applications by restricting write permission on the master model.

Create a new master model drawing

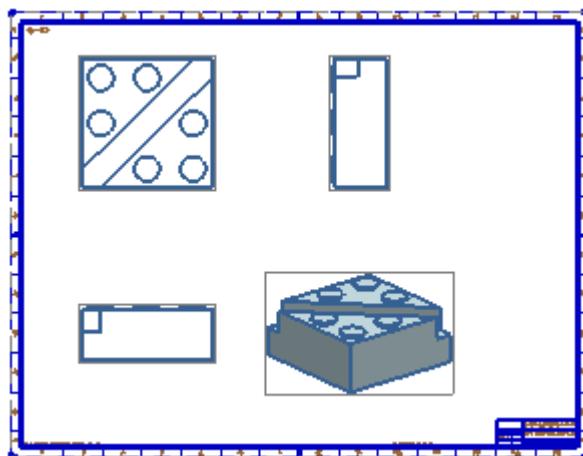
This example shows how to create a non-master drawing from a master part.



1. Open your part and select **File→New**.
2. Click the **Drawing** tab.
3. On the **Drawing** page, select an appropriate drawing template.
 When **Units** is set to millimeters or inches, unit-specific templates will display. Make sure to select a template that has the same units as the current part.
4. (Optional) In the **Name** box, type a name.
5. (Optional) Choose a new folder location by clicking the **Browse** button next to the **Folder** box.

6. Click **OK** to create the new drawing part.

In this example, we created an E sized template and added views.

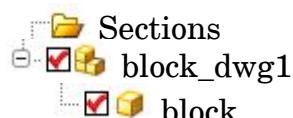


When you create a drawing this way, by default the template will set all of your drafting preferences to those contained in the template.

7. Click the **Assembly Navigator**  tab in the Resource bar.

Note that you are now working in an assembly file and the original part file has been added as a component.

Descriptive Part Name



Although this is an assembly of the original part, all Drafting commands and operations perform in the same way as they do in the original part.



Sheet overview

Use the **Sheet** command to:

- Create a new drawing in your work part, if no drawing exists in the part.
 Use the **File→New** command and select a template from the **Drawing** tab if you want to create and save the drawing in a separate part.
- Create a new sheet if a drawing already exists in the part.
- Edit an existing sheet.

From the **Sheet** command you can:

- Create drawing sheets from a template.
- Create custom sized sheets.
- Set the units and projection angle for standard sheet sizes.
- Set the height, width, units, and projection angle for custom sheet sizes.
- Edit the size, scale, projection angle, units, and name of an existing sheet.
 You can change the projection angle only if there are no projected views on the drawing sheet.

Where do I find it?

To create a new sheet

Application	Drafting
Toolbar	Drawing→New Sheet 
Menu	Insert→Sheet
Part Navigator	Right-click the Drawing node® Insert Sheet

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To edit an existing sheet

Application	Drafting
Toolbar	Drafting Edit→Edit Sheet 
Menu	Edit→Sheet
Shortcut menu	Right-click the drawing border ® Edit Sheet
Part Navigator	Right-click the Sheet node® Edit Sheet

Create a new drawing sheet

1. On the **Drawing** toolbar, click **New Sheet** 
2. In the **Sheet** dialog box, define the drawing sheet size, scale, name, units of measure and projection angle.
3. Choose **OK**.

Open a drawing sheet

Do one of the following:

- In the **Part Navigator**, double click the drawing sheet node.
- In the **Part Navigator**, right-click the drawing sheet node and choose **Open**.
- On the **Drawing** toolbar, click **Open Sheet** 

Edit a drawing sheet

Do one of the following:

- In the **Part Navigator**, right-click the drawing sheet and choose **Edit Sheet**.
- Right-click the view border of a drawing sheet and choose **Edit Sheet**.
- On the **Drafting Edit** toolbar, click **Edit Sheet** 
- From the menu, choose **Edit→Sheet**.



You can change the projection angle only if no projected views exist on the drawing sheet.

You can edit the drawing sheet to a larger or smaller size. If you edit the drawing sheet to a size so small that a member view falls entirely outside the boundary of the drawing sheet, you will get an error message.



If you need to edit the drawing sheet to a smaller size, but cannot due to the current position of the views, move the views closer to the drawing sheet's origin at the lower left corner of the sheet.

Delete a drawing sheet

Do one of the following:

- Right-click the border of a drawing sheet and choose **Delete**.
- In the **Part Navigator**, right-click the drawing sheet node and choose **Delete**.
- Choose **Edit ® Delete**, then select the current drawing sheet border, then click **OK**.

Change drawing display to monochrome

The **Monochrome Display** option displays a drawing sheet in a single color.

1. Choose **Preferences® Visualization**.
2. Click the **Color Settings** tab.
3. In the **Drawing Part Settings** section, select the **Monochrome Display** check box.

The default colors are black and gray. You can specify any line or background color



In the **Part Navigator**, right-click the drawing node and choose **Monochrome**.

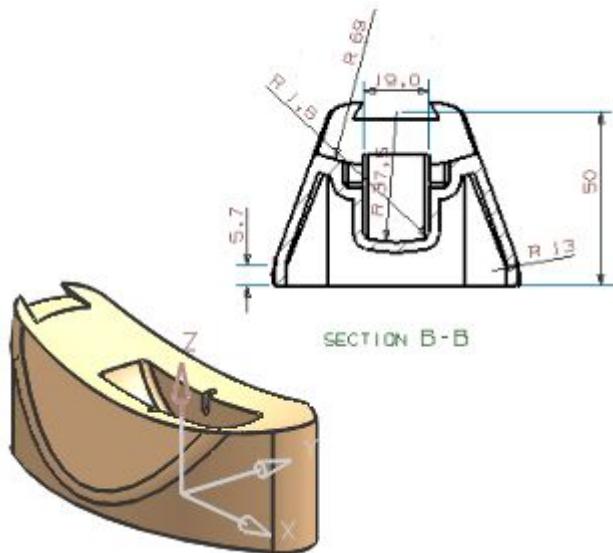
Monochrome will be applied to all drawing sheets in the part.

In the **Visualization Preferences** dialog box, on the Line page, use the **Show Widths** option to display of line widths and make the display closely resemble a plotter output.

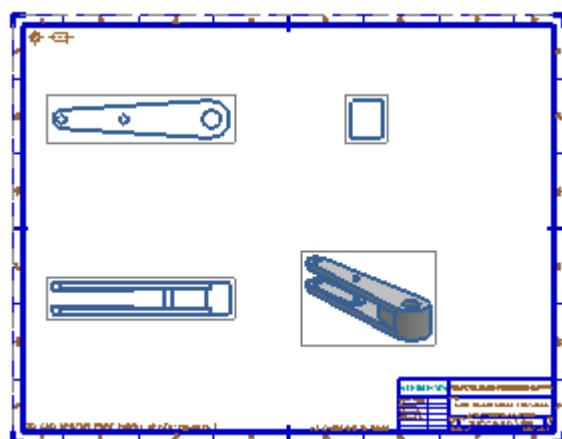
Activities: Drafting – Edit a master model, Create drawings

In the *Introduction to Drafting* section, do the activities:

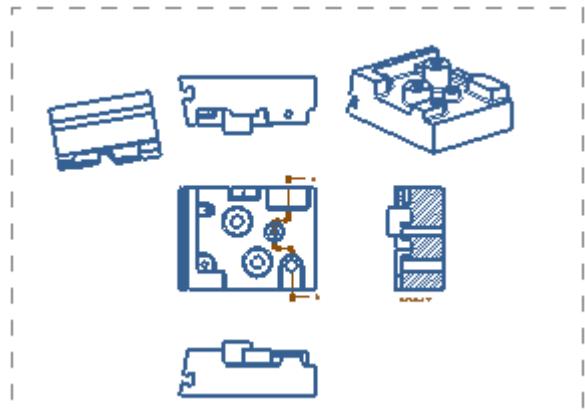
- (Optional) *Edit parameters of a master model.*



- *Create a new non-master drawing*



- Open and edit drawings



Drafting View Style/View Preference overview

The dialog box is used to create or modify the visual appearance of a view, as well as control the preference setting for the style options associated with all view types.

- The dialog box is called the **View Style** dialog box if you are creating or modifying a specific view's appearance.
- It is called the **View Preference** dialog box if you are setting the global preferences for all view types.

The title of the dialog box, the tabs displayed in the dialog box, and the options displayed on the different tabs of the dialog box, depend on whether you are setting the style of a view or setting the preference for all views.

From this dialog box you can:

- Control visual properties of the view such as how hidden lines, visible lines, virtual intersections, and smooth edges are rendered, and whether the view is displayed as a shaded or wireframe image.
- Set the perspective, angle, and scale of the view.
- Control the visibility of the view annotation such as the view label, scale label, and **centerline** symbols.
- Set the display characteristics for **tracelines** and thread representations in the view.
- Control the appearance of a **section view**, including the display of foreground, background, and **crosshatch** elements.

Where do I find it?

To open the **View Style** dialog box

Application	Drafting
Toolbar	Drafting Edit→Edit Style  , then select one or more views from the drawing sheet
Radial Toolbar	Highlight the view border, press and hold the right mouse button → Style 
Menu	Edit→Style then select one or more views from the drawing sheet or Part Navigator
Graphics window	Double-click a view boundary
Part Navigator	Right-click one or more view boundaries ® Style

To open the **View Preferences** dialog box

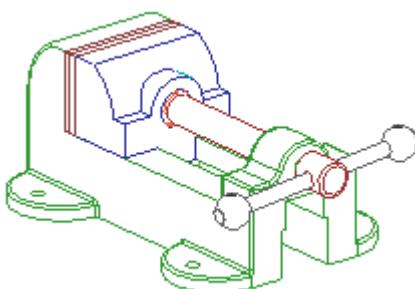
Application	Drafting
Toolbar	Drawing→View Preferences 
Menu	Preferences→View

Hidden Lines overview

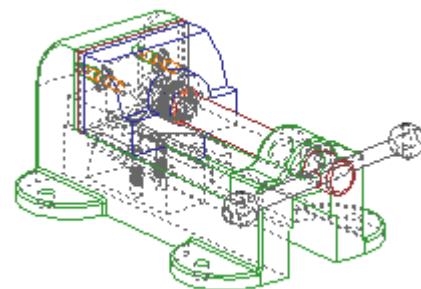
The **Hidden Line** tab in the **View Style** and **View Preferences** dialog box provides options for controlling the way edges and curves, hidden by geometry based on the view orientation, are displayed.



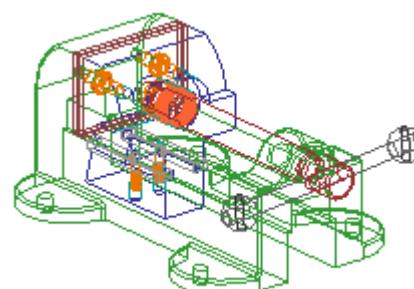
Any view created from an existing view, known as the parent view, will automatically inherit the parent view's hidden line settings, regardless of how the **Hidden Line** options in the **View Style** or **View Preferences** dialog boxes are set. Once the view is created, however, you can change its hidden line display using the **View Style** dialog box.



Hidden edges and curves are invisible



Hidden edges and curves are displayed with specific color and font



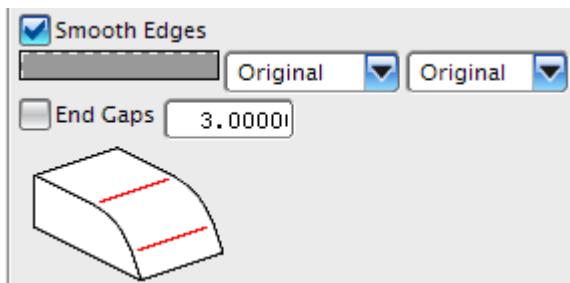
All hidden edges and curves are displayed

Smooth Edges

Smooth edges are those whose adjacent faces have the same surface tangent at the edge where they meet.

On the Smooth Edges page, select the **Smooth Edges** option to use the color, font, and width settings to specify the appearance of smooth edges.

Use the **End Gaps** option to vary the edge intersection appearance.





Base overview

Use the **Base** command to add any standard modeling or custom view saved in a part to a drawing sheet. A single drawing sheet may contain one or more base views. From base views, you can create associated child views such as **Projected**, **Section**, and **Detail** views.

The **Base** command provides options that enable you to:

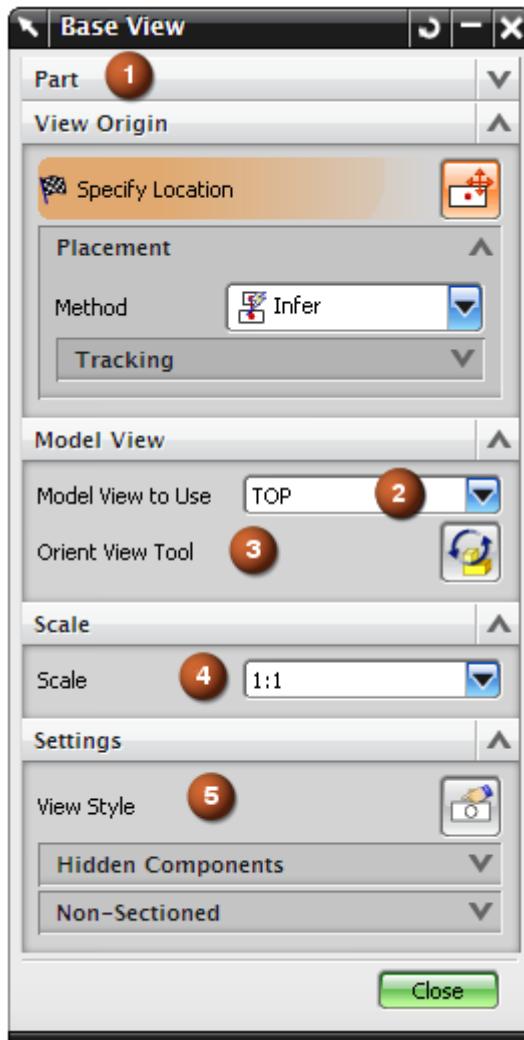
- Add any view from the current part or another loaded part.
- Specify the position and orientation of a view on the drawing.
- Set the scale and style of the view.
- Control the appearance of components in views on assembly drawings.

Where do I find it?

Application	Drafting
Toolbar	Drawing ® Base View
Menu	Insert→View→Base
Shortcut Menu	Right-click a drawing sheet border® Add Base View
Part Navigator	Right-click a sheet node® Add Base View

Base View options

- (1) **Part** Add a view from a part that you specify
- (2) **Model View to Use** Select the base view type from a list. Select NX defined views or custom views.
- (3) **Orient View Tool** Define a custom orientation for a view such as perpendicular to a model face.
- (4) **Scale** Select from a list of several preset scales, enter a custom scale, or define the scale by an expression.
- (5) **View Style** Opens the **View Style** dialog box. Settings you make apply to the view you are adding.



Create a Base view

1. Choose **Start→Drafting**.



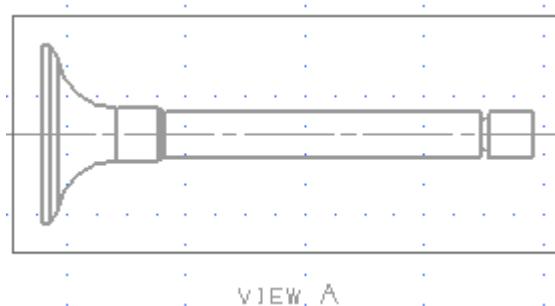
2. On the **Drawing** toolbar, click **Base View**, or choose **Insert→View→Base**.
3. In the graphics window, move your cursor to the desired location and right-click.

The shortcut menu is displayed.

4. Select **Model View to Use® Right** from the list.
5. Right-click again and select **View Label**.
6. Click in the graphics window to place your view.

7. Click middle mouse button to dismiss the **Base View** dialog box.

The right model view is displayed with a label below the view.





Projected overview

You can project views from an existing **base**, **drawing**, orthographic, or **auxiliary** view. NX automatically infers orthographic and auxiliary alignment as you move the cursor in a circular motion about the parent view's center.



The **Automatically Start Projected View Command** option on the **General** tab of the **Preferences→Drafting** dialog box controls the automatic start up of the projected view command.

The system automatically infers:

- A **hinge line** to use as a reference to rotate the view into orthographic space.
- A vector direction that is perpendicular to the hinge line. The arrow indicates the projection direction from the parent view.

You can manually define the hinge line and also reverse the projection direction before you place the view.

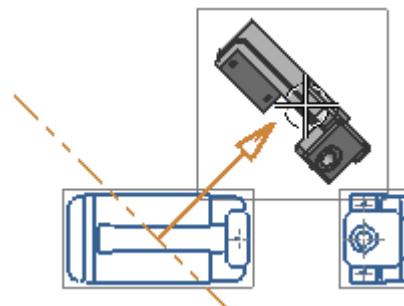
Where do I find it?

Application	Drafting
Toolbar	Drawing ® Projected View
Menu	Insert→View→Projected
Shortcut Menu	Right-click a view border® Add Projected View
Part Navigator	Right-click a view node® Add Projected View

Projection lines

When you move the cursor while adding a projected view you see projection lines. You can place the view at any angle from the base view. You can:

- Place the view manually. The angle snaps to 45° increments.
- Define a hinge line.
- Select a planar face and project perpendicular to it.



Preview

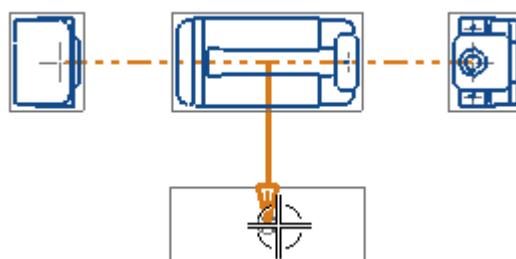
As you move the cursor the preview style can be:

- Border
- Wireframe
- Hidden Wireframe
- Shaded Image



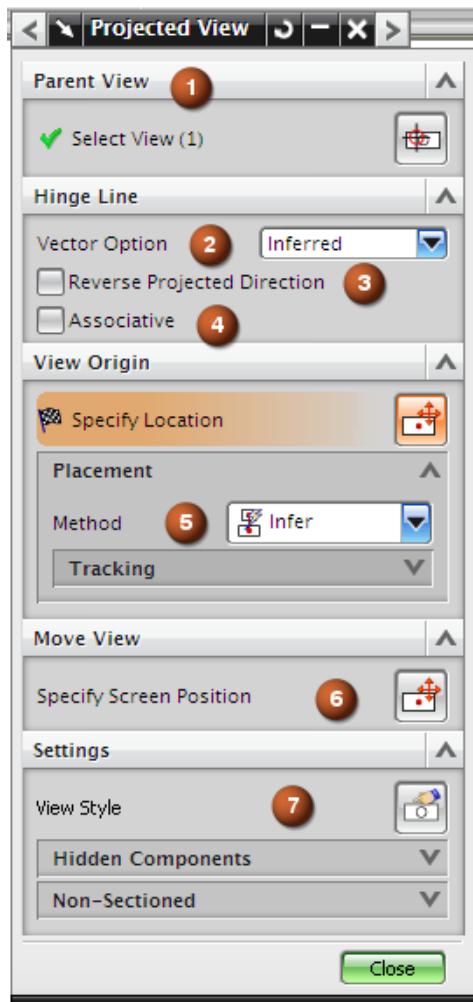
To select a preview option, right-click before you place the view and choose **Preview Style**.

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Projected View options

- (1) **Parent View** Select a different base view to use as the parent view.
- (2) **Vector Option** Infer a hinge line or explicitly define a fixed hinge line.
- (3) **Reverse Projected Direction** Reverse the direction of the projected view.
- (4) **Associative** Make the projected view associated to the defined hinge line.
- (5) **Placement** Align the projected view horizontally, vertically, perpendicular to the hinge line, or infer the placement based on the cursor location.
- (6) **Move View** Move an existing view without interrupting the interaction to place a projected view.
- (7) **View Style** Open the **View Style** dialog box.



Edit the style of an existing view

There are several ways to change the style of an existing view.

- Double-click the view border.
- Right-click the view border and choose **Style**.
- In the **Part Navigator**, double-click a drawing view node.
- In the **Part Navigator**, right-click a drawing view node and choose **Style**.
- Choose **Edit→Style**.

Drag views on a drawing

1. (Optional) Select one or more views to move.
2. Hold the cursor over the border of a view (a *selected* view, if there are more than one) until it changes to drag mode .
3. Drag the view as required.



As you move a view relative to others, alignment lines appear. When you place a view with alignment lines visible it automatically snaps to an aligned position.

Delete views on a drawing

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There are several ways to remove a view from a drawing sheet.

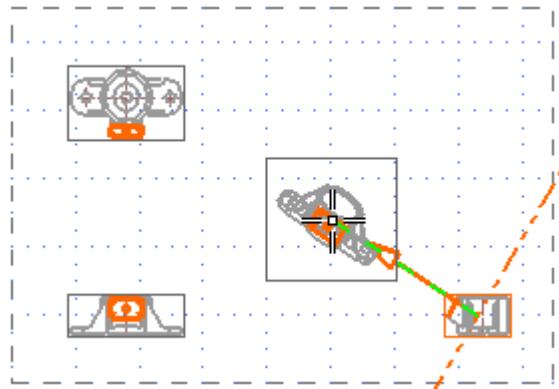
- Right-click the view border and choose **Delete**.
- In the **Part Navigator**, right-click the view to be removed and choose **Delete**.
- Choose **Delete**  and select the view.
- Choose **Edit→Delete** and select the view.

Once a view is removed from a drawing sheet, all drafting objects or view modifications associated to that view are deleted.

Activities: Drafting — add views

In the *Introduction to drafting* section, do the activity:

- *Add views to a drawing*



Dimensions

The Drafting application supports ANSI, ASME, ISO and JIS standard dimension types, including Baseline, Chain, and Ordinate dimensions.

- NX uses an intelligent inference algorithm to anticipate and create dimensions based on the objects you select.
- You can also choose to create specific dimension types using the **Dimension** toolbar, or from the **Insert→Dimension** menu.
- You can set local preferences that control the display of the dimension type.

Create a dimension — general procedure

A general procedure for creating a dimension is as follows:

1. (Optional) Before creating any dimension, you can set the local preferences for dimension objects by choosing **Preferences ® Annotation**, or clicking  **Annotation Preferences** on the **Annotation** toolbar.
2. In the Drafting application, choose **Insert ® Dimension** and select the dimension type you want to create, or on the **Dimension** toolbar, select a dimensioning option from the **Drafting Dimension Drop-down** list.
3. Select the object(s) you want to dimension. Use the Line and Snap Point options (when available) to help with selection.
4. (Optional) While rubber banding the dimension, you can use right-click options to control the display and placement of the dimension.
5. Indicate a position for the dimension origin by clicking the left mouse button.



Annotation Preferences

Use the **Annotation Preferences** dialog box (**Preferences→Annotation**) to configure global settings that affect dimensions.

The following pages in the **Annotation Preferences** dialog box apply to dimensions:

Dimensions Control the display of extension lines and arrows, orientation of text, precision and tolerance, chamfer dimensions, and narrow dimensions.

Line/Arrow Control the style and size of leaders, arrows, and extension lines for both dimensions and other annotations. A preview area provides a rendition of the symbol with leaders and dimensions.

Lettering Control the alignment, justification, size, and font of text.

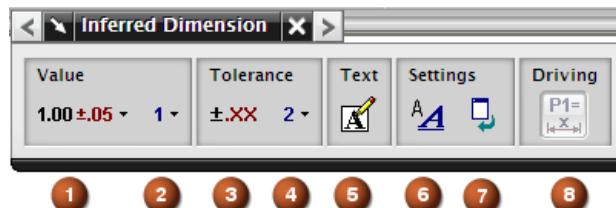
Units Control the desired unit of measure for dimensions and whether dimensions are created in single or dual dimension format.

Radial Control the settings that are unique to diameter and radius dimensions.

Dimension preferences and placement

When you select a dimension type, the corresponding dimension dialog bar appears.

The settings that you set on the dialog bar affect only dimensions you are currently creating. The settings return to global values when you exit dimension creation or choose **Reset**.



1 Tolerance Types	Select the tolerance type from a list.
2 Primary Nominal Precision	Select the primary nominal precision from 0-6 decimal places from a list. If the preferences format is fractional, then the list displays fractional precision values.
3 Tolerance Values	Enter a tolerance value, or values, using on-screen input boxes.
4 Tolerance Precision	Set the primary tolerance precision from 0-6 decimal places.
5 Text Editor	Display the full Text Editor dialog box where you can enter symbols and appended text.
6 Dimension Style	Open the Dimension Style dialog box. Use this option to affect settings as you create one or more dimensions. The global settings are restored when you exit from creating dimensions.
7 Reset	Reset local preferences to previous current settings in the part and clear appended text.
8 Driving Dimension	Treat as a driving sketch dimension or as a documentation dimension. This option is only available when a sketch is created on the drawing sheet.

Annotation placement options

When you select a dimension type to create, the annotation placement options appear on the Selection bar.



1 Leader Orientation	Set the leader on the left side, right side, or automatically infer the side.
2 Associative Origin	Associate the entity origin so that it is always aligned with another dimension.
3 Alignment Position	Specify the alignment position on object such as top-left, mid-center, bottom-right.
4 Origin Tool	Open the Origin Tool dialog box.

Snap Point options

Snap point options appear on the Selection bar while you are working with dimensions.

These options act as a filter for selecting geometric points. You can either select or deselect any of these in order to limit your selection to specific types of points.

Use the **Two-curve Intersection** button (at the right end of the toolbar) to select any two edges whose intersection you cannot fit inside the select ball. When you select it, all the other buttons are unavailable.

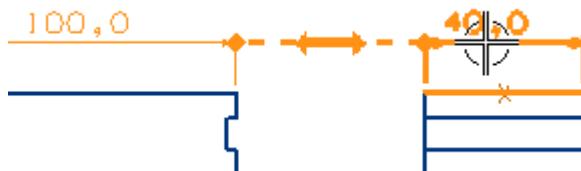


You can press the Esc key at any time to release all selected objects.

Placement cues for dimensions

As you create dimensions, you can align them with an existing dimension. Graphical cues appear when the origins of two dimensions are vertically or horizontally aligned.

If you want the new dimension associated with the existing dimension, make sure the **Associative Origin** button is active.



Append text to a dimension

You can append text to a dimension while you are creating it.

If you want only one line of appended text, select the object(s) to dimension and, before you place the dimension, choose one of the appended text options in the shortcut menu.



You may also press the right (after), left (before), up (above), or down (below) arrow key on the keyboard to open a window to input a single line of text.

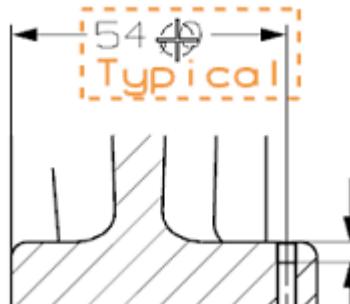


If the text is complex, use the **Text Editor** from the dimension dialog bar.

To add appended text to a previously created dimension that does not already have appended text, do one of the following:



- Use **Note** to create the text, then drag the text until it's highlighted in a dashed box in the correct placement position.



- Double-click the dimension, and open the **Text Editor** from the dialog bar.
- Double-click the dimension, and use the shortcut menu to choose either **Appended Text** (for a single line of text), or **Text Editor** (for complex text).
- Double-click the dimension, and use the Right (after), Left (before), Up (above), or Down (below) arrow key on the keyboard to get the appended text location you desire. Type the text and press Enter.

To edit existing appended text, do one of the following:

- Double-click the appended text.
- Double-click the dimension and use the Right (after), Left (before), Up (above), or Down (below) arrow key on the keyboard to get the appended text location you desire.
- Select the dimension, and open the shortcut menu over the appended text.

Change text orientation and text arrow placement

- To set the text orientation and text arrow placement as you create a dimension, open the shortcut menu before you place the text.
- To change the text orientation and text arrow placement of an existing dimension, edit the dimension style.

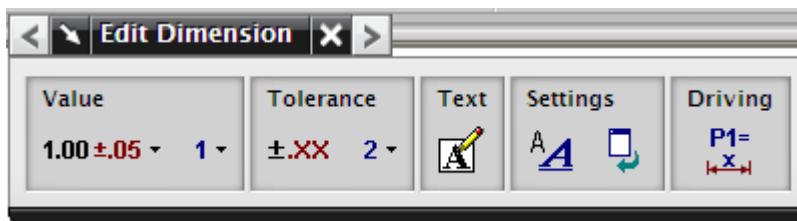
Move a dimension

To change the origin of an existing dimension, simply drag it when no command is active.

The cursor will change to  when you are in the move mode.

Edit a dimension

1. Use one of the following methods for selecting a dimension.
 - Select a dimension. Right-click, choose the appropriate option, and execute your edit.
 - Double-click a dimension.
This action selects the dimension and activates the relevant dimension dialog bar for editing.
 - Select a dimension. Choose **Edit ® Annotation ® Text**.
 - Choose **Edit ® Annotation ® Text**, then select a Dimension.
-  Once you select a dimension for editing, you are in Edit mode, as indicated by the wrench cursor  and the presence of the **Edit Dimension** dialog bar.



2. Edit the selected dimension or select another dimension to edit.
To select another dimension, either click or double-click another dimension, and based on where you selected, an on-screen input box may also display.
3. Click Esc or middle mouse button to deactivate the dimension icon options when you are finished editing.

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Change the precision of a dimension

1. Double-click the dimension.
2. Do one of the following:
 - Choose **Nominal Precision** from the shortcut menu.
 - From the **Edit Dimension** dialog bar, in the **Value** group, click the precision list.
 - On the keyboard, press the number key that corresponds to the desired precision.

Inherit preferences from an existing dimension

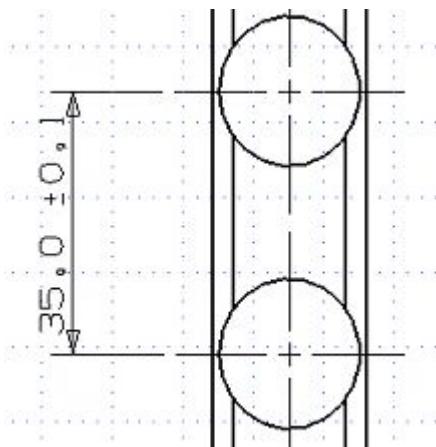
After you create a dimension, you can edit its preference settings to match another dimension:

1. Double-click the dimension you want to change.
2. Right-click the dimension and choose **Inherit**.
3. Select the dimension that has the desired preference settings.

Activities: Drafting — dimensions

In the *Introduction to drafting* section, do the activity:

- *Create dimensions*





Note overview

Use the **Note** dialog box to create and edit notes and labels. A note consists of text; a label consists of text with one or more **leader lines**.

Text can be imported by reference to expressions, part attributes and object attributes, and can include symbols formed from control character sequences or user-defined symbols.



While editing or creating notes, labels, or GD&T, NX provides a preview directly in the graphics window as you enter each character.

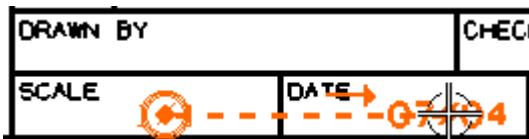
Where do I find it?

Application	Drafting
Toolbar	Annotation→Note
Menu	Insert→Annotation→Note

Helper lines

Helper lines act as a guide to allow you to align notes, labels, dimensions, symbols, and views with other objects on the drawing sheet. Helper lines appear as a dashed line.

To use helper lines, move the cursor over the object to which you want to align as you are placing the new annotation. The note highlights and helper lines appear.



Click to place the annotation at the desired location.

Create a note



On Windows, while in the Drafting application, you can drag a text file onto a drawing sheet to create a note.

1. Open the **Note** dialog box in one of these ways:

- Choose **Insert→Annotation→Note**.

- On the **Annotation** toolbar, click **Note** .

2. Type the desired text in the Text Input box.

Text appears in the text box and at your cursor location in the graphics window.

3. Move the cursor to the desired location and click to place the note.

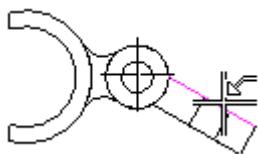
Create a label

1. On the **Annotation** toolbar, click **Note**  or choose **Insert→Annotation→Note**.

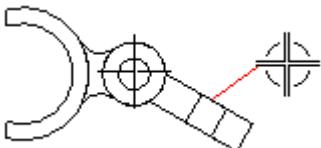
2. Type the desired text into the Text Input box.

Text appears in the text box and in the graphics window.

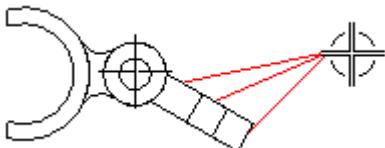
3. Place the cursor over geometry.



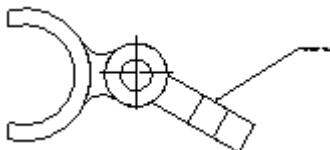
4. Click and drag to create a leader.



To create multiple leaders, click and drag on different geometry.



5. Click again to place the label on the drawing (single leader shown below).



Tips

- Before placing the note, you can select additional options from the **Note** dialog box.
- You can align annotations using helper lines that appear when you move the cursor near existing annotation.

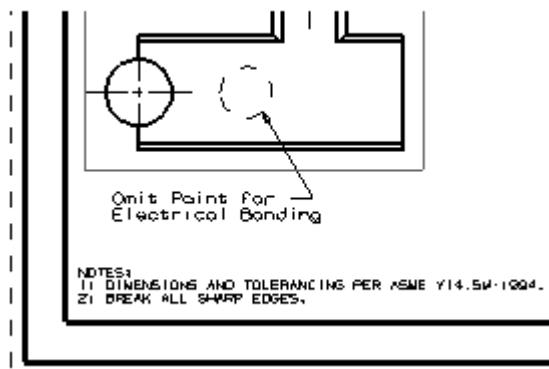
Edit an existing note or label

1. Right-click the existing note or label.
 2. Choose the appropriate option from the shortcut menu.
-  You can display the Note dialog box and edit text by double-clicking the note or label.

Activities: Drafting — Notes and labels

In the *Introduction to drafting* section, do the activity:

- *Create notes and labels*



Summary: Drafting

Use the Drafting application to create and edit drawing sheets. Views and dimensions on a drawing sheet are associative to the solid model and update when changes are made to the model.

Use the **Note** command to create notes and labels.

In this lesson you:

- Applied the master model concept to create a drawing.
- Modified a drawing sheet.
- Added views to a drawing sheet.
- Created dimensions.
- Added notes to a drawing sheet.

Lesson

28 Editing models

Purpose

This lesson introduces various tools to edit and interrogate a model.

Objectives

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

- Replay model construction.
- Measure the distance between objects.
- Assign a material to a solid body and calculate mass properties.
- Move, replace, and resize faces.
- Delete faces of a solid body to remove internal detail.

Feature Replay overview

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

You can:

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



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

Where do I find it?

Toolbar	
Prerequisite	A feature must appear in the Part Navigator in order to step to it with Feature Replay .
Menu	Tools® Update® Make First Feature Current Make Previous Feature Current Make Next Feature Current Make Last Feature Current Make Next Boolean Current Automatic Feature Replay

Feature and object information

The **Information** menu offers a number of options to obtain information about the model.

Information→Feature

- Opens the **Feature Browser** dialog box where you can identify parent/child relationships between a selected feature and the other features in the model.
 - When the **Display Dimensions** check box is selected, expressions that control the feature are displayed in the graphics window.
 - Click **OK** or **Apply** to display the Information window with the geometric data and associated expressions.
-  Feature information may also be accessed by selecting the feature in the **Part Navigator** and choosing **Information** from the shortcut menu, or by selecting the feature in the graphics window and choosing **Properties** from the shortcut menu.

Information→Object

- Displays information about selected objects such as name, layer, color, object type, and geometric properties (length, diameter, start and end coordinates, etc.) in an Information window.
- Any type of geometric object may be selected including curves, edges, faces, and bodies.

Information→Expression→List All

- Lists all expressions in the part in the Information window.
- From the Information window, you can print the listing or save it as a text file.

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Information→Expression→List All by Reference

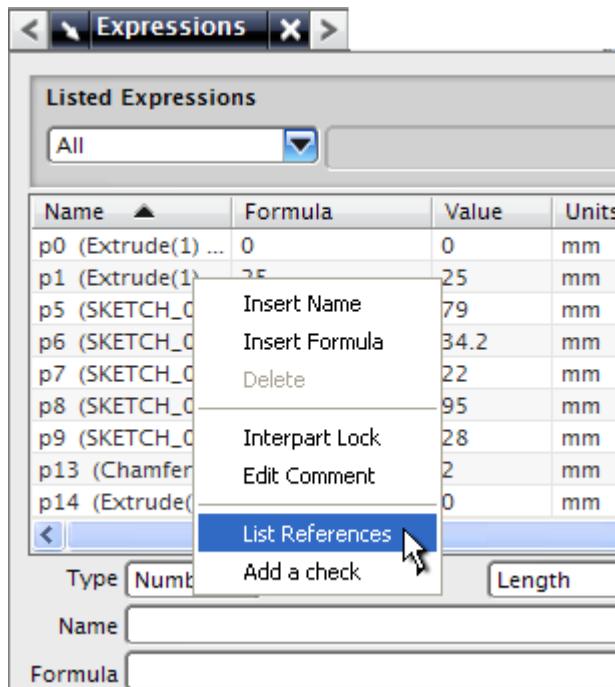
- Identifies expressions that reference other expressions and the features that they define.
- You can use the **Edit→Find** option on the Information window menu bar to search for a specific expression.

Referenced expressions

If an expression defines a feature directly, the feature name is listed with it in the **Expressions** dialog box.

Any expression can be *referenced* by the formula of other expressions.

You can identify all referencing expressions by using **List References** in the shortcut menu.



List referenced expressions

1. Choose **Tools**→**Expression**.
2. If necessary, change the **Listed Expressions** filter to list the expression.
3. Right-click the expression and choose **List References**.

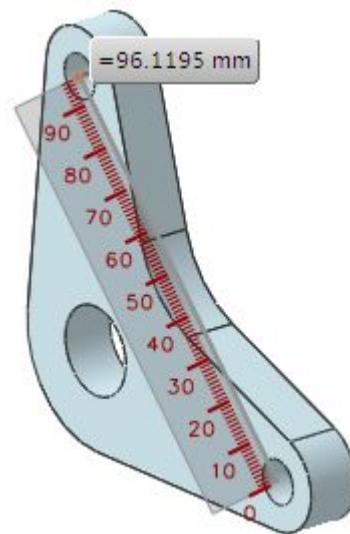
An Information window lists the features and other expressions that reference the selected expression.



Measure Distance overview

Use the **Measure Distance** command to obtain the distance between any two objects such as points, curves, planes, bodies, edges, faces, or components.

To specify units for distance measurements, choose **Analysis→Units**.



Where do I find it?

Application	Gateway
Toolbar	Utility→Measure Distance
Menu	Analysis→Measure Distance

Find the minimum distance between two objects



1. On the **Utility** toolbar, click **Measure Distance** or choose **Analysis→Measure Distance**.
2. In the **Measure Distance** dialog box, from the **Type** list, select **Distance**.
3. In the **Measurement** group, from the **Distance** list, select **Minimum**.
4. Select the first point or object.
5. Select the second point or object.

A temporary ruler and measurement result are displayed in the graphics window.

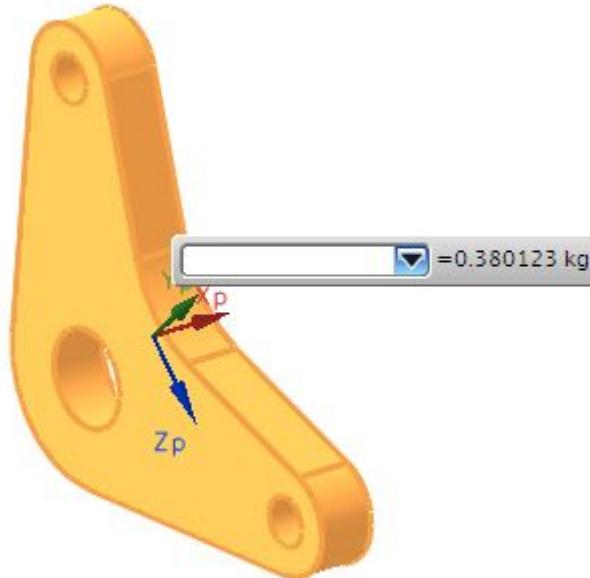


In the **Results Display** group, select the **Show Information Window** check box to display result details in the Information window.

Measure Bodies overview

Use the **Measure Bodies** command to calculate the volume, mass, surface area, radius of gyration, weight, and centroid of selected bodies.

To specify units for body measurements, choose **Analysis→Units**.



There are two ways to assign a density to a solid body.

- Choose **Tools→Material Properties** and assign a material.
- Choose **Edit→Feature→Solid Density**.

The default density is specified in Modeling Preferences.

Where do I find it?

Application	Gateway
Toolbar	Active Mockup® Analysis Drop-down list® Measure Bodies
Menu	Analysis→Measure Bodies

Assign a material to a solid body

1. Choose **Tools→Material Properties**.

The materials in the NX Material Library are listed by default. You may create additional custom materials.

2. In the graphics window, select the solid body.
3. If the material is not listed in the **Assign Material** dialog box, expand the **Filters** list and search for the material by name, category, or type.
4. In the **Materials** dialog box, select a material from the **Materials** list.
5. Click **OK**.

Delayed updates

As you add features to your model, it may take noticeably longer to update.

You can delay updates until after edits are made.

From the main menu, choose **Tools**→**Update**→**Delayed after Edit**, or, on the **Edit Feature** toolbar, click .

- If **Delayed Update after Edit** is *inactive*, the part is updated after the completion of each edit operation. This is the *default* setting.
- If **Delayed Update after Edit** is *active*, feature updates are delayed while edits are made.

When **Delayed Update after Edit** is active and edits are made, **Update Model** is available.

Choose **Tools**→**Update**→**Update Model**, or, on the **Edit Feature** toolbar, click .

 The model is updated automatically when the part is saved.

Synchronous Modeling

Synchronous Modeling commands are used to modify a model regardless of its origins, **associativity**, or feature history.

The model you modify can be:

- Imported from other CAD systems
- Non-associative, with no features
- A native NX model complete with features

By working directly with the model, the geometry is not rebuilt or converted.

With Synchronous Modeling, designers can use parametric features without the limitations of a feature history.

Synchronous Modeling is primarily suited for use on models composed of analytic faces types like plane, cylinder, cone, sphere, torus. This does not necessarily mean “simple” parts, since models with many thousands of faces are composed of these face types.



Move Face overview

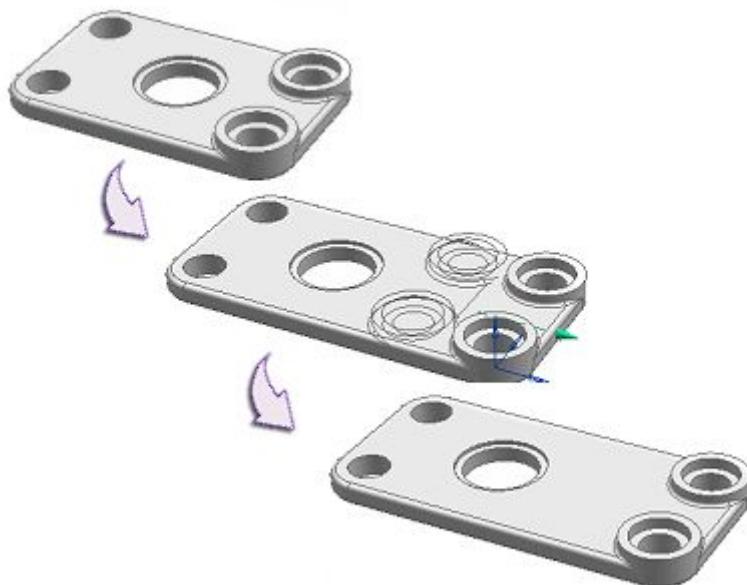
Use the **Move Face** command to move a set of faces and automatically adjust adjacent blend faces.

You can use linear or angular transform methods to move the selected faces.

Move Face is a useful design tool that facilitates easy design change during the design process. It is also useful in downstream applications like Tooling, Manufacturing, and Simulation, where you can directly make changes to the model, regardless of feature history.

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

- To relocate a group of faces to a different position to meet design intent.
- To relocate a series of faces in several components of an assembly. (All components and assembly must be in History-Free mode.)
- To change the bend angle of a sheet metal part that has no history.
- To rotate a face or set of faces about a given axis and about a point. For example, to change the angular position of a keyway slot.
- To change the orientation of an entire solid body, irrespective of its history, to a different orientation.

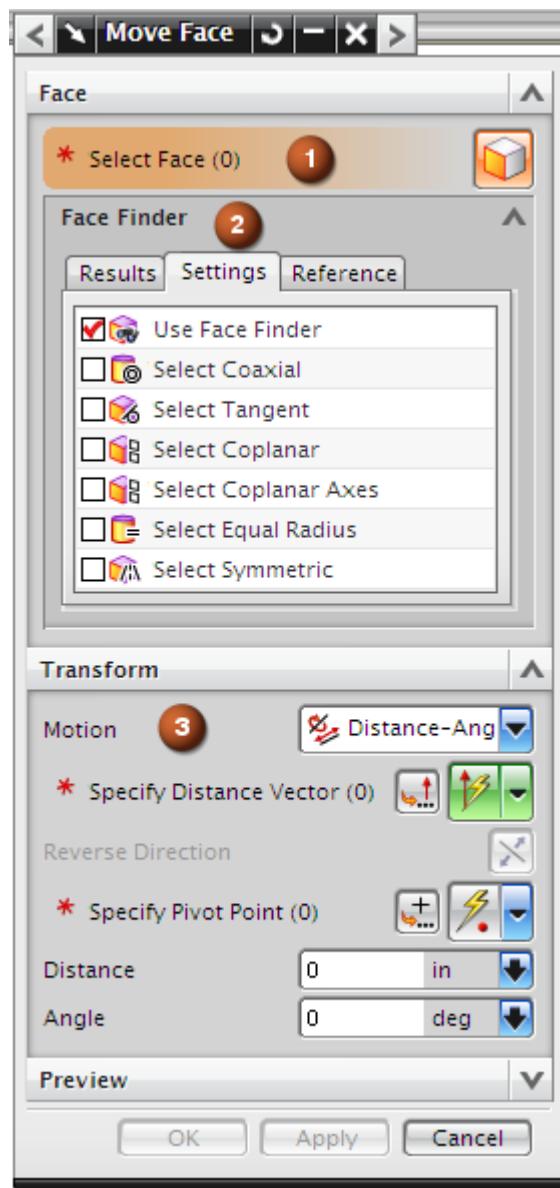


Where do I find it?

Application	Modeling and Shape Studio
Toolbar	Synchronous Modeling® Move Face 
Menu	Insert® Synchronous Modeling® Move Face

Move Face options

- | | |
|----------------------|--|
| 1 Select Face | Lets you select a face or faces to move. |
| 2 Face Finder | Lets you select faces based on how their geometry compares with the selected face. |
| 3 Motion | Provides linear and angular transform methods for the faces you select to move. |

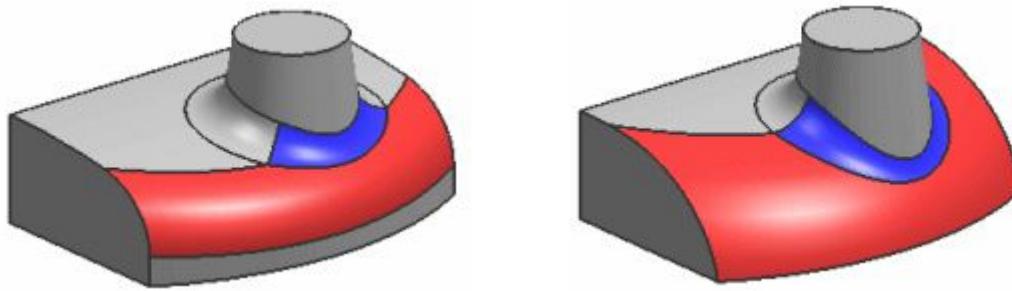




Resize Blend overview

Use the **Resize Blend** command to change the radii of **blend** faces regardless of their feature history.

The **Resize Blend** command works with translated files and unparameterized solids.



In this example, the red face in the body on the left is resized using **Resize Blend**.

The dependent blue face updates automatically.

Where do I find it?

Application	Modeling
Toolbar	Synchronous Modeling® Resize Blend
Menu	Insert® Synchronous Modeling® Resize Blend



Replace Face overview

Use the **Replace Face** command to change the geometry of a face, such as making it simpler or replacing it with a complex surface.

You can:

- Replace a set of faces with one or more faces. The replacement face or faces are typically from a different body but can also be from the same body as the face to replace. The selected replacement faces must be on the same body and form an edge-connected chain. If these conditions are not met, NX displays an error.
- Replace solid faces or sheet faces.
- Automatically reblend adjacent blends when the replacement face is a single face.
- Extend the replacement face to form a complete intersection with the body.
- Offset the replacement face for the eventual replacement face.

You can use **Replace Face** even on nonparameterized models.

Where do I find it?

Application	Modeling, Shape Studio, Manufacturing
Toolbar	Synchronous Modeling® Replace Face 
Menu	Insert® Synchronous Modeling® Replace Face

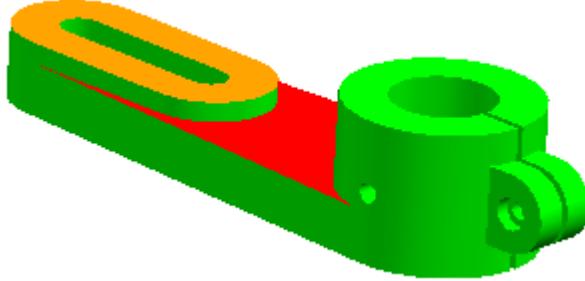


Replace a single face with another face

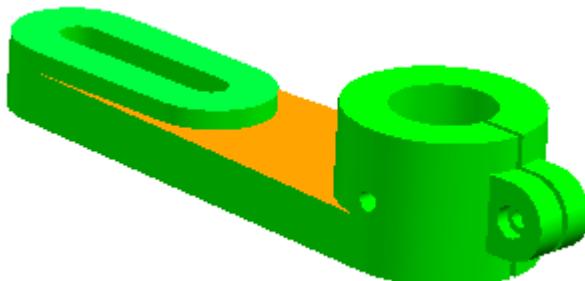
1. On the **Synchronous Modeling** toolbar click **Replace Face** or choose **Insert® Synchronous Modeling® Replace Face**.

In the **Replace Face** dialog box, **Select Face** is active in the **Face to Replace** group.

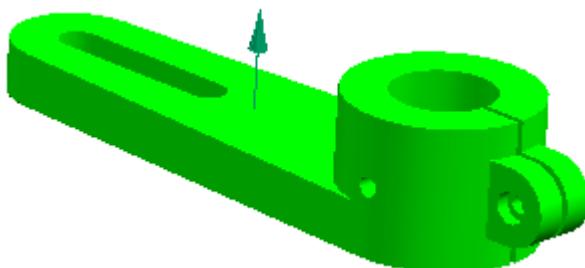
2. Select the face to replace.



3. In the **Replacement Face** group, click **Select Face** and select the replacement face.



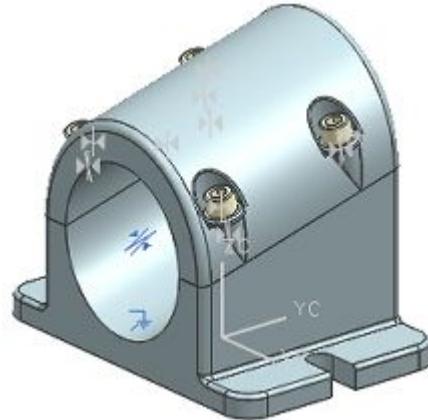
4. (Optional) Click **Reverse Direction** to reverse the displayed direction of the replacement face.
5. Click **OK** or **Apply** to replace the selected face with the replacement face.



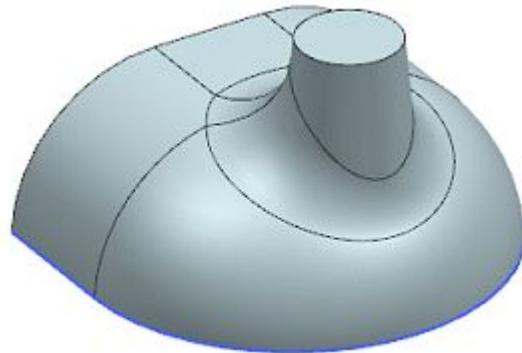
Activities: Move and replace faces

In the *Editing models* section, do the activity:

- *Face and Synchronous Modeling operations*



- *Synchronous Modeling – Resize Blend*





Delete Face overview

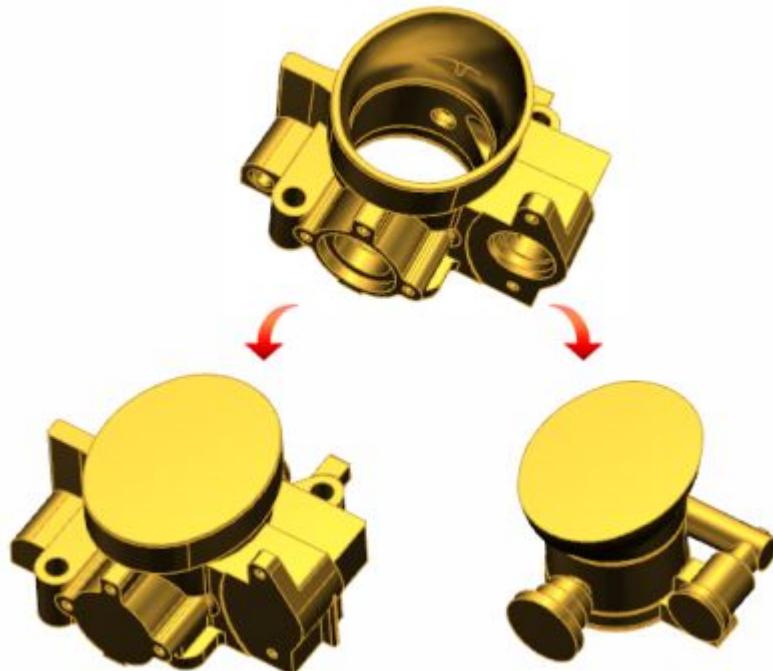
Use the **Delete Face** command to delete faces.

You can:

- Automatically heal the open area left in the model by the deleted faces, by extending adjacent faces.
- Preserve adjacent blends.

In the **History Modeling** mode, after you delete a face, the delete face feature appears in the history of the model. You can edit or delete this like any other feature.

Delete Face is especially useful when modifying an imported model which has no feature history.



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Internal faces deleted

External faces deleted



Delete Face replaces the former **Simplify Body** command.

Where do I find it?

Application	Modeling, Shape Studio, Manufacturing
Toolbar	Synchronous Modeling
Menu	Insert® Synchronous Modeling® Delete Face

Delete Face uses

Delete Face has the following uses:

Assembly Performance

Detail can be removed from an extracted solid in a component part to reduce the amount of data loaded in an assembly. The simplified solid can provide an accurate representation without unnecessary detail.

Internal Volume Solid

A solid representing the internal volume of a part can be created by removing external faces and retaining internal faces. This solid can be analyzed for volume and mass properties.

Core and Pattern Preparation

Delete Face can often be used both to remove interior faces, for patterns, and to remove exterior faces, for cores.

In-Process Parts

Faces can be removed from a machining solid to represent the part as-cast, or, as it would appear at various stages in a manufacturing process.

Remove Proprietary Data

A solid can be modified to remove proprietary detail before it is exported to a supplier or subcontractor. The modified solid can be linked into another part. The linked part would have no "knowledge" of removed data in the original, but it can still be updated by the owning company if the parent body changes.

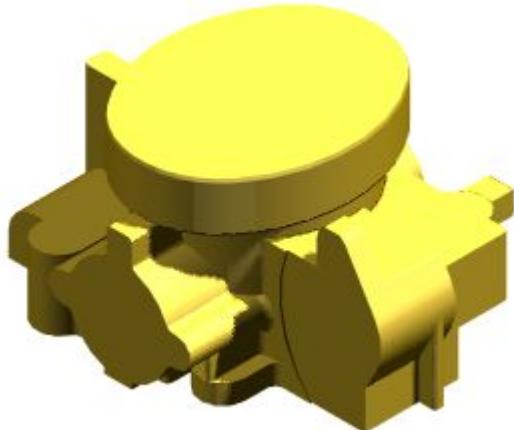
Finite Element Analysis

Details such as holes and blends can be removed for finite element analysis.

Activities: Delete Face

In the *Editing models* section, do the activities:

- *Delete faces of a solid*



Summary: Editing models

In this lesson you:

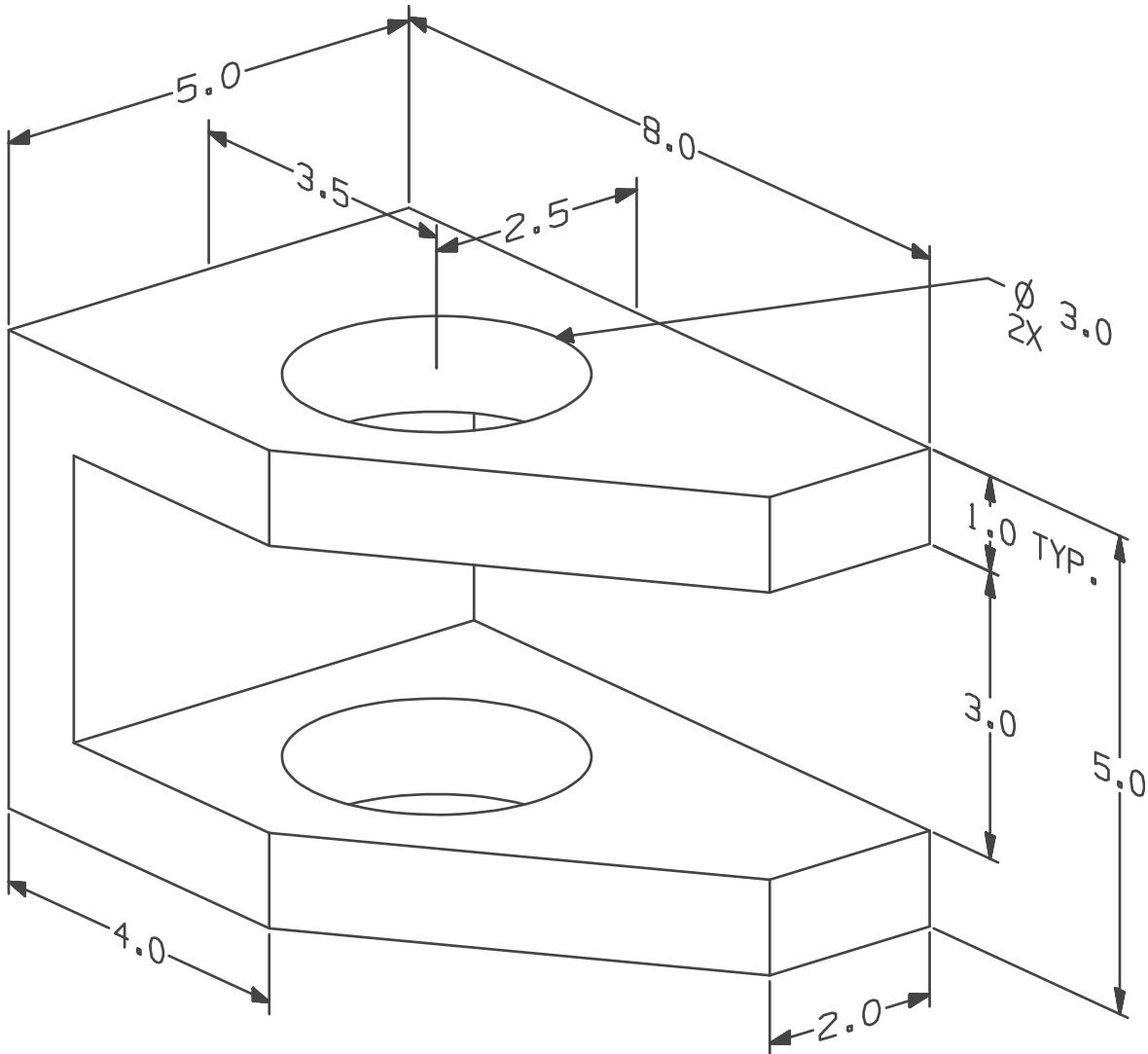
- Replayed model construction.
- Measured the distance between objects.
- Assigned a material to a solid body and calculated mass properties.
- Used Synchronous Modeling commands to move, replace and delete faces.

Appendix

A Practice projects

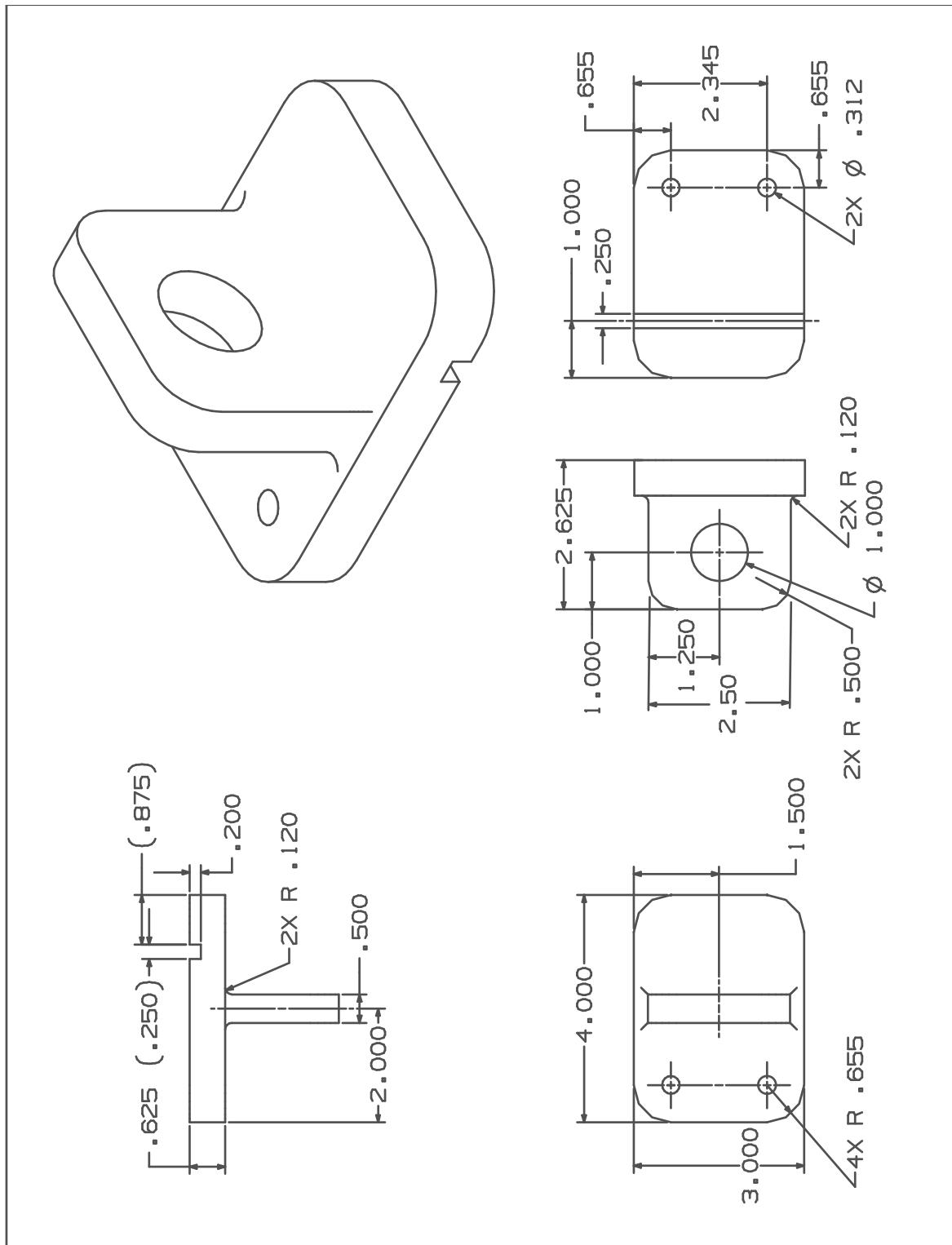
This appendix contains additional practice projects.

Practice Project 1



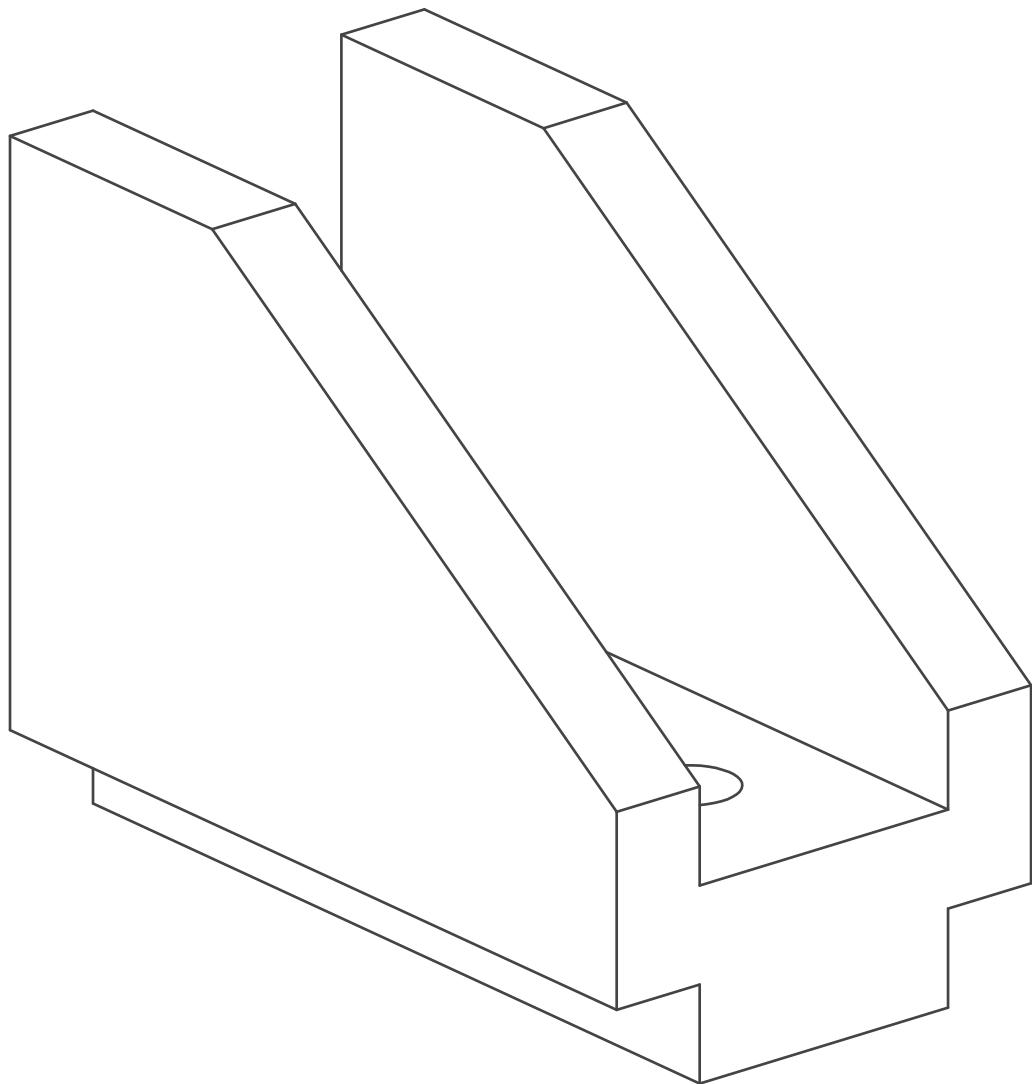
A

Practice Project 2

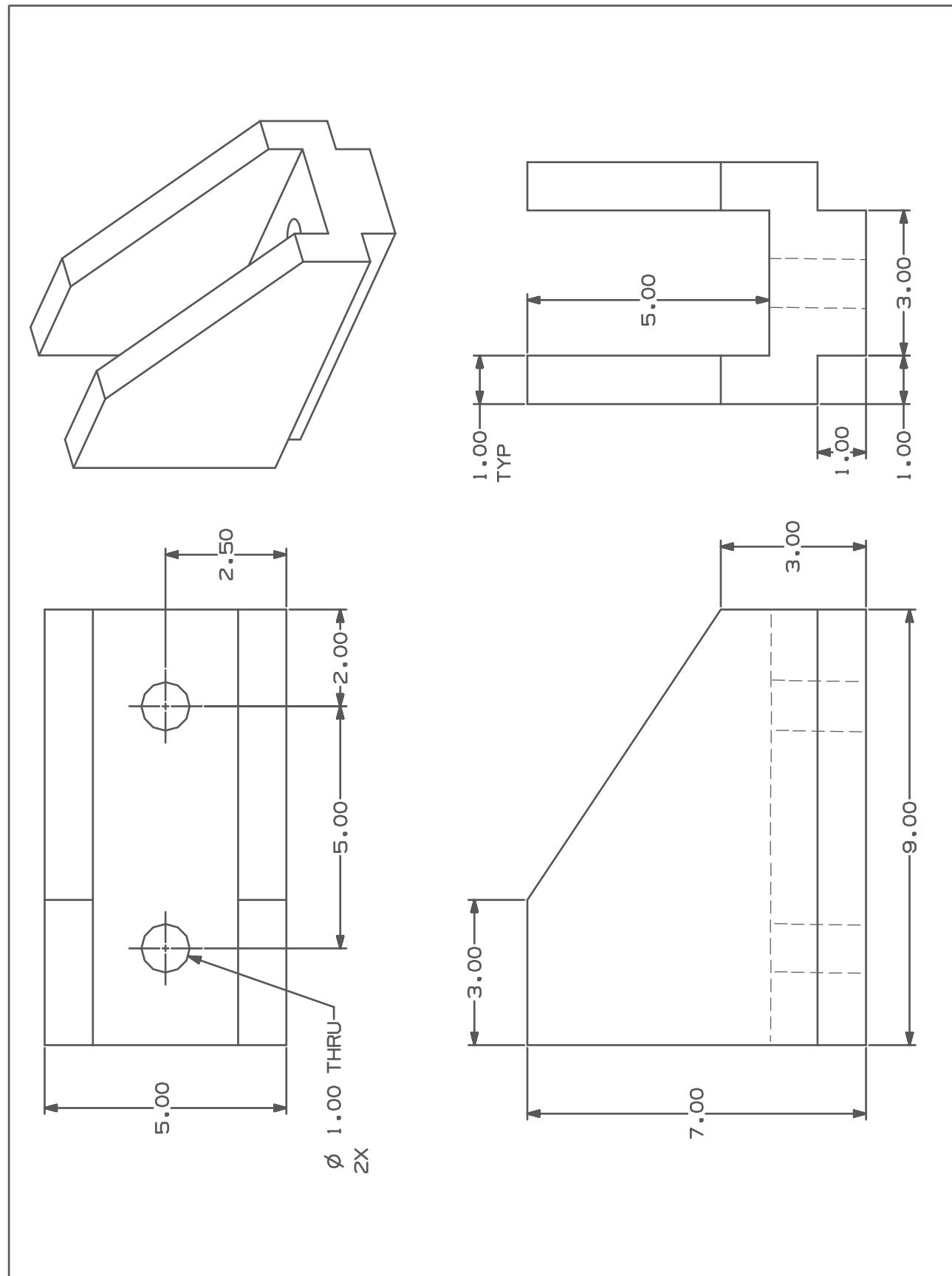


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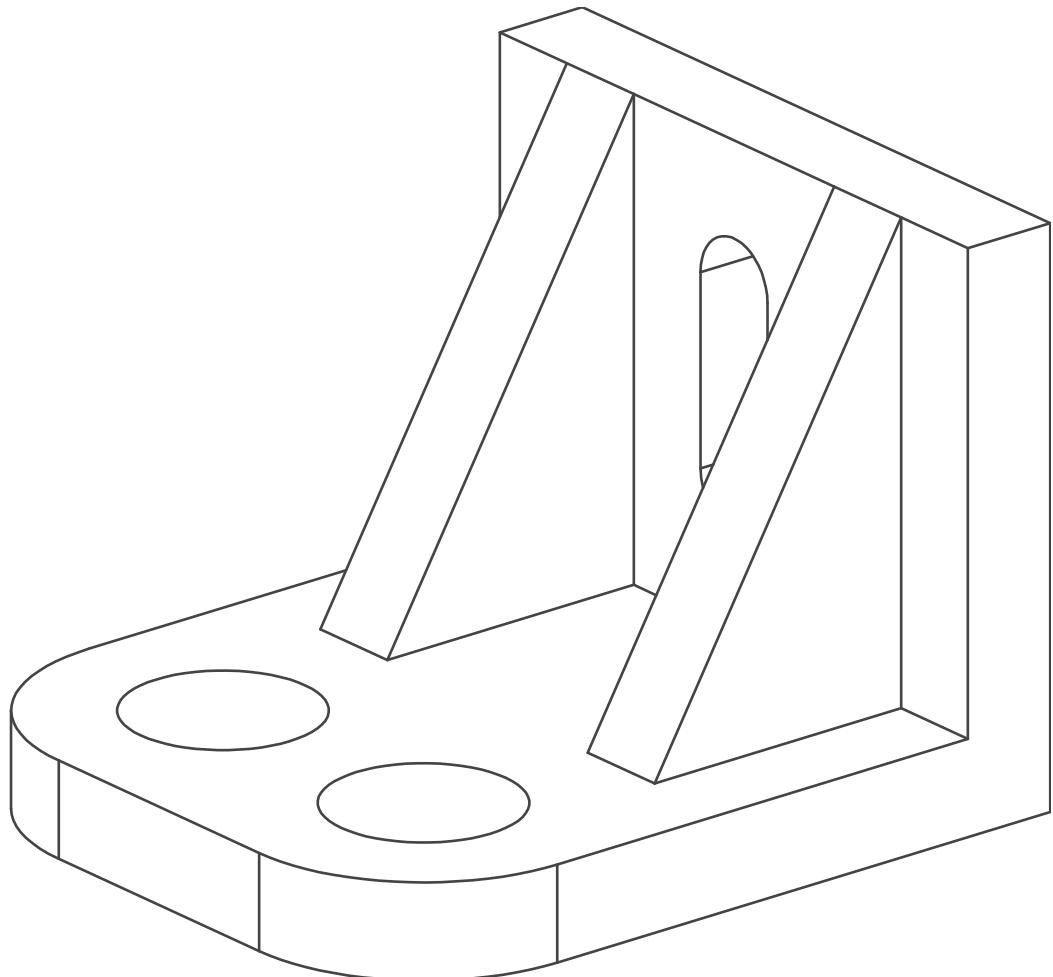
Practice Project 3



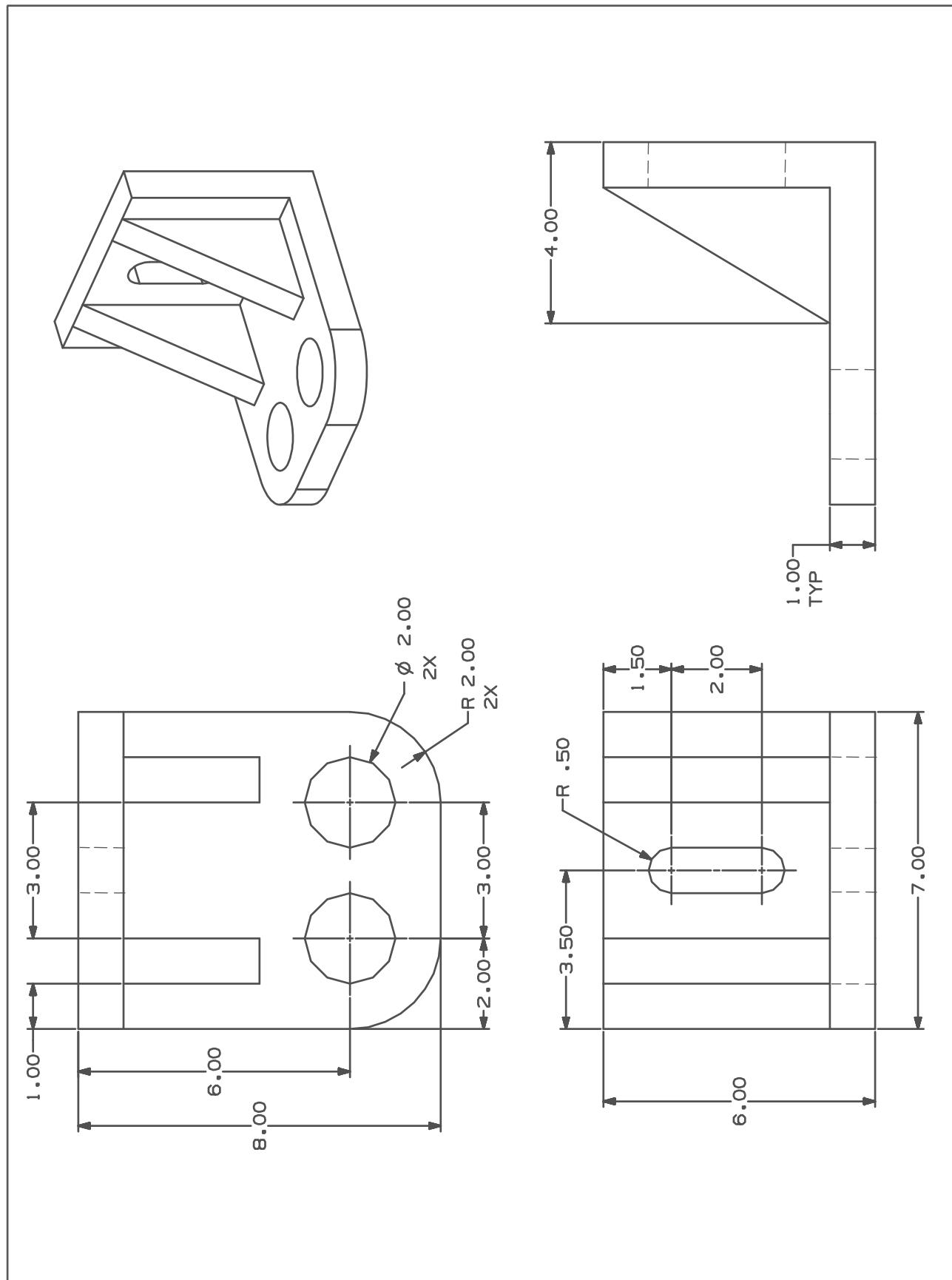
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Practice Project 4

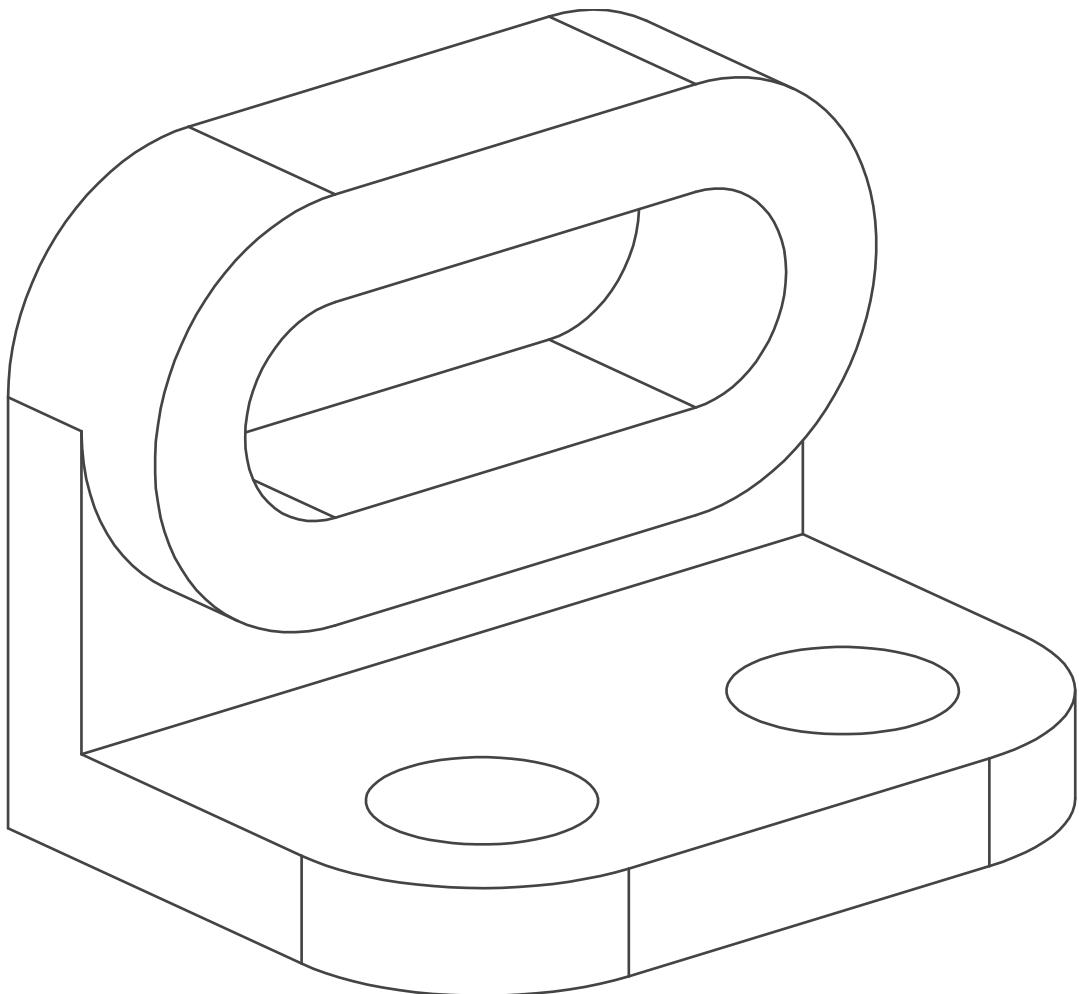


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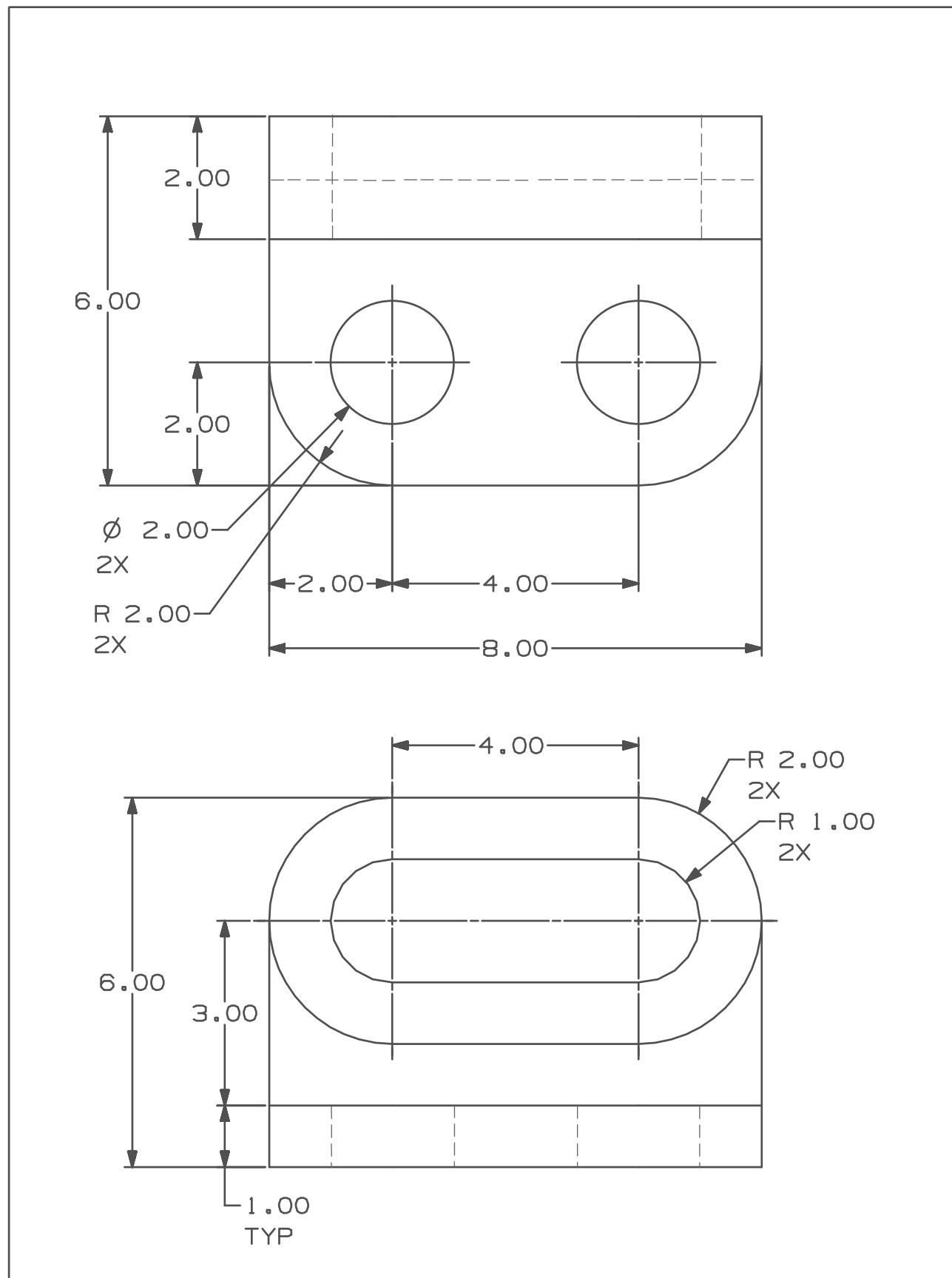


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Practice Project 5

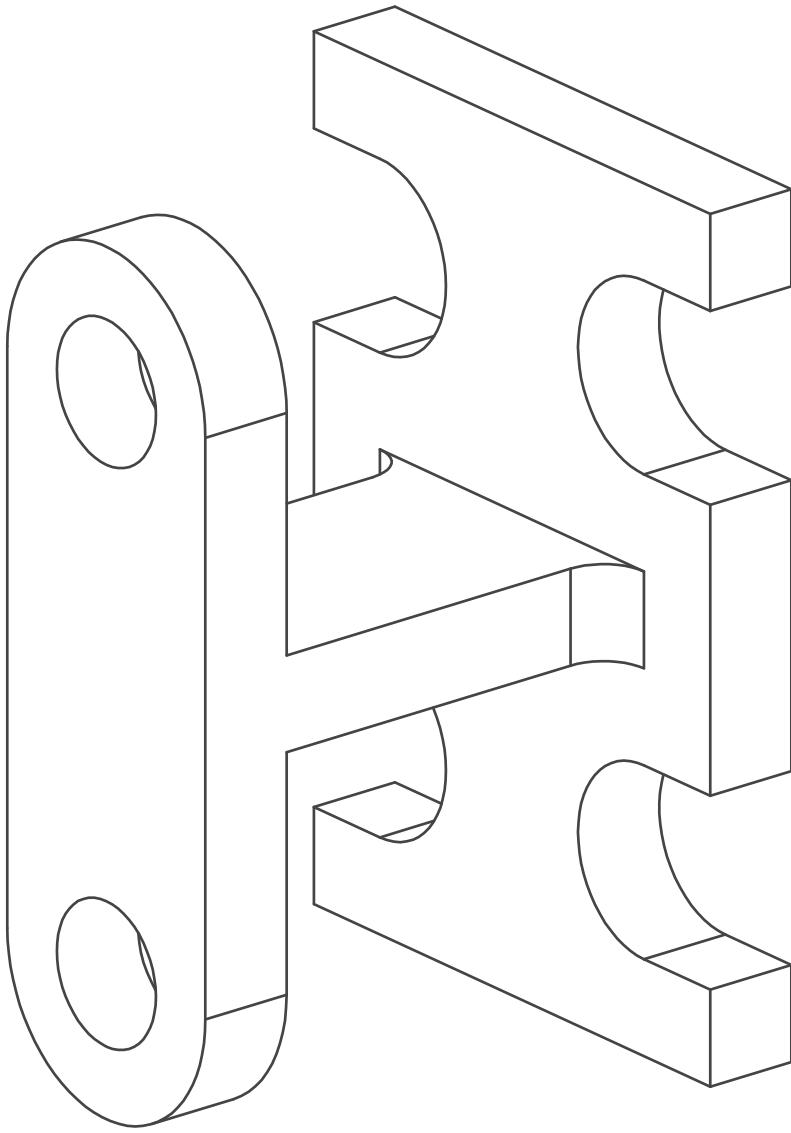


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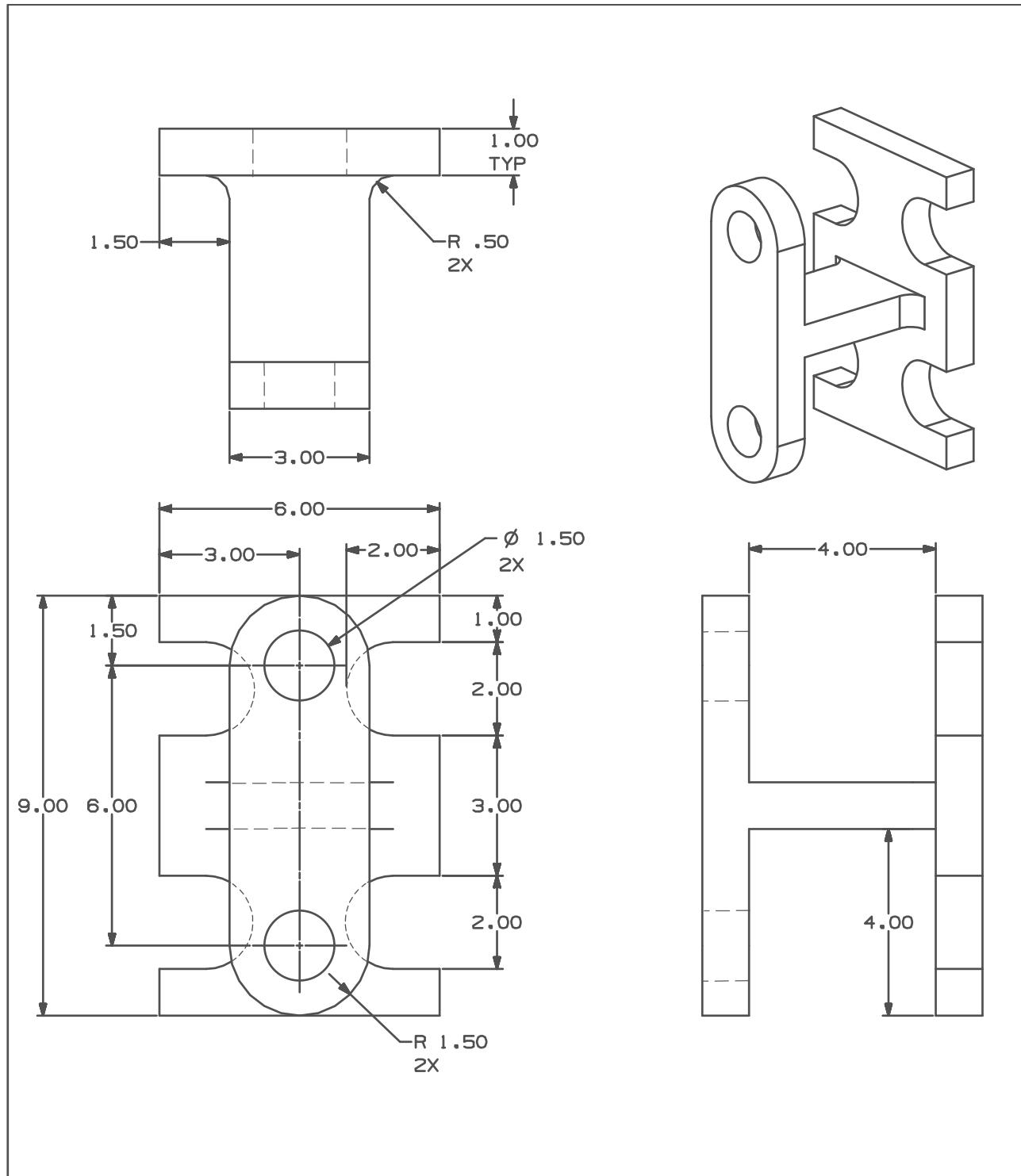


A

Practice Project 6

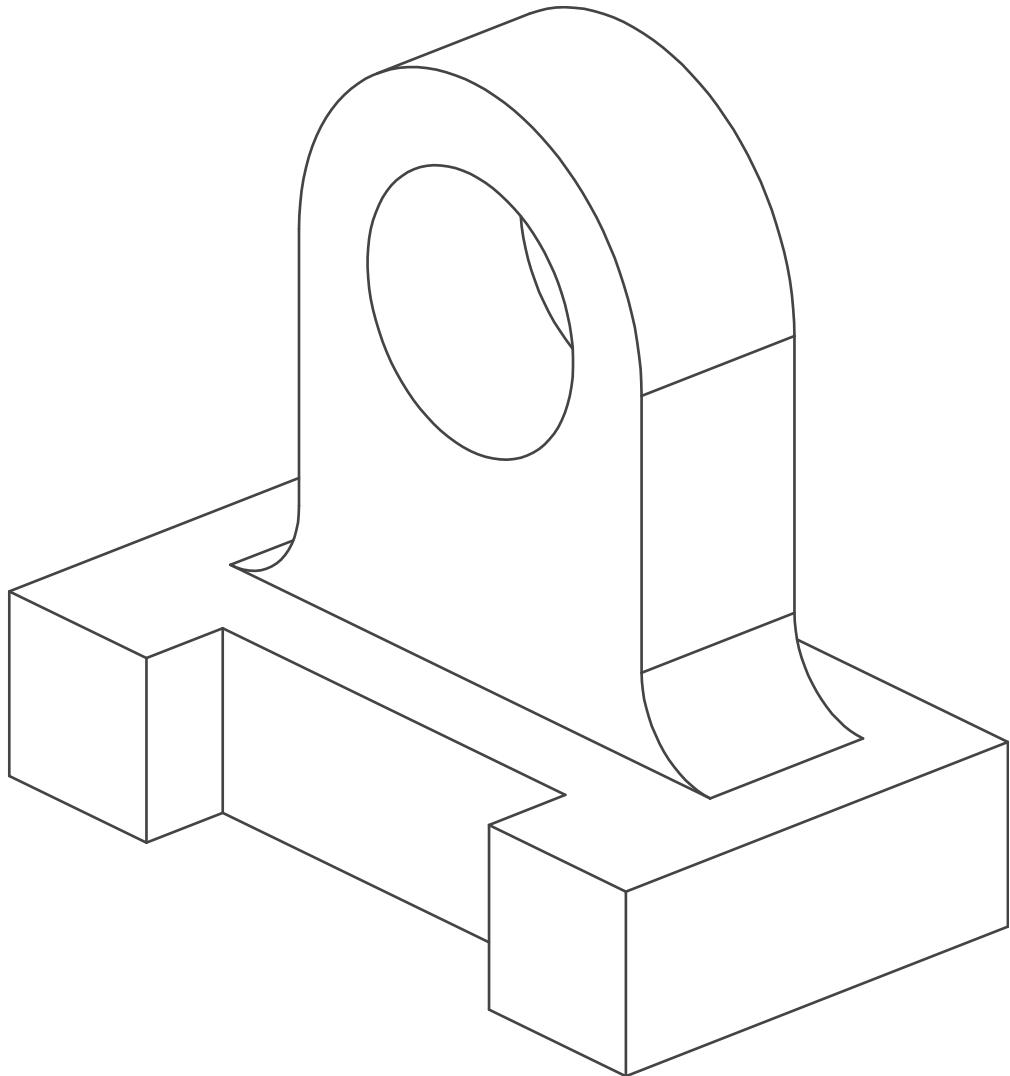


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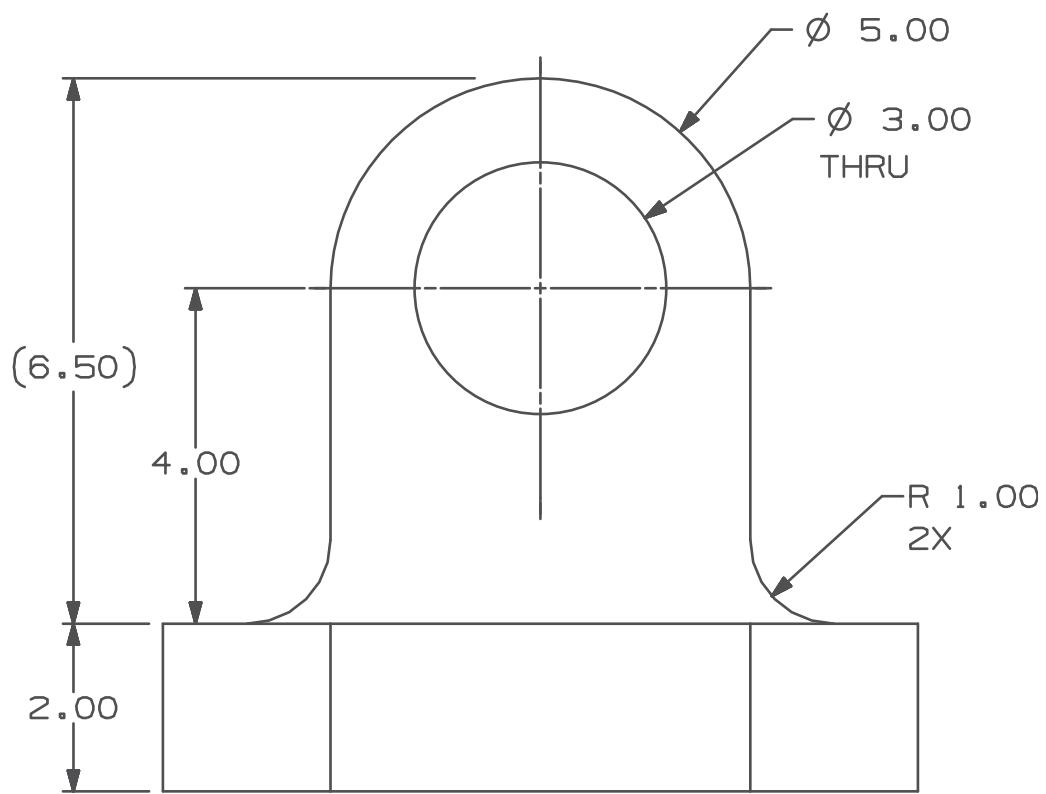
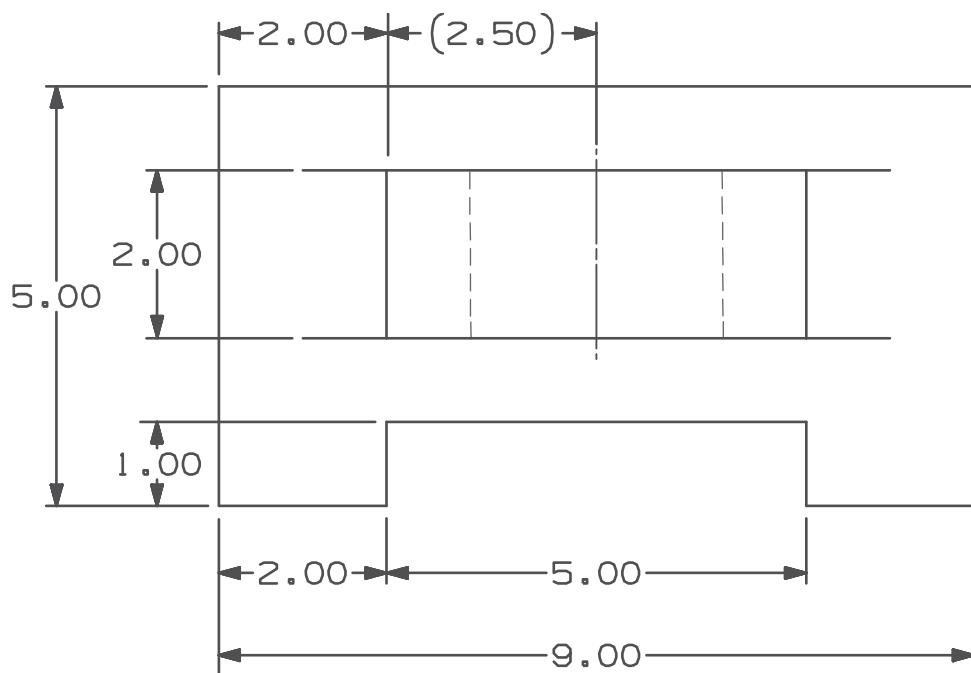


A

Practice Project 7

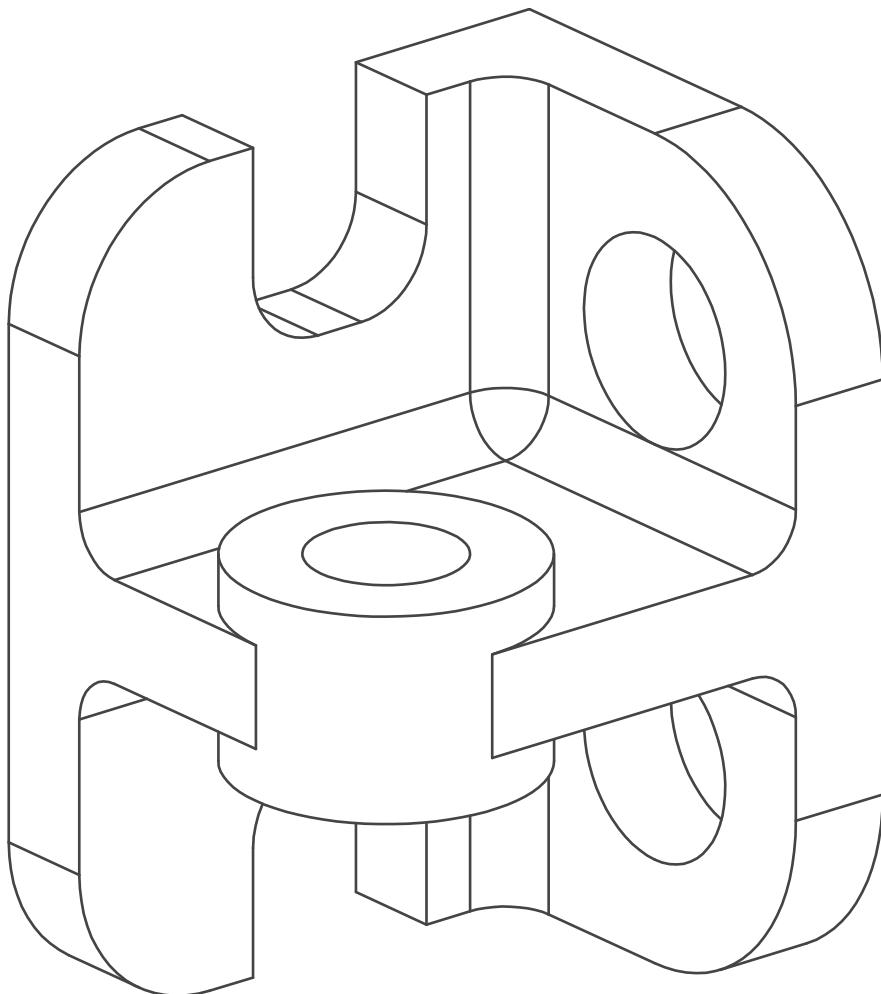


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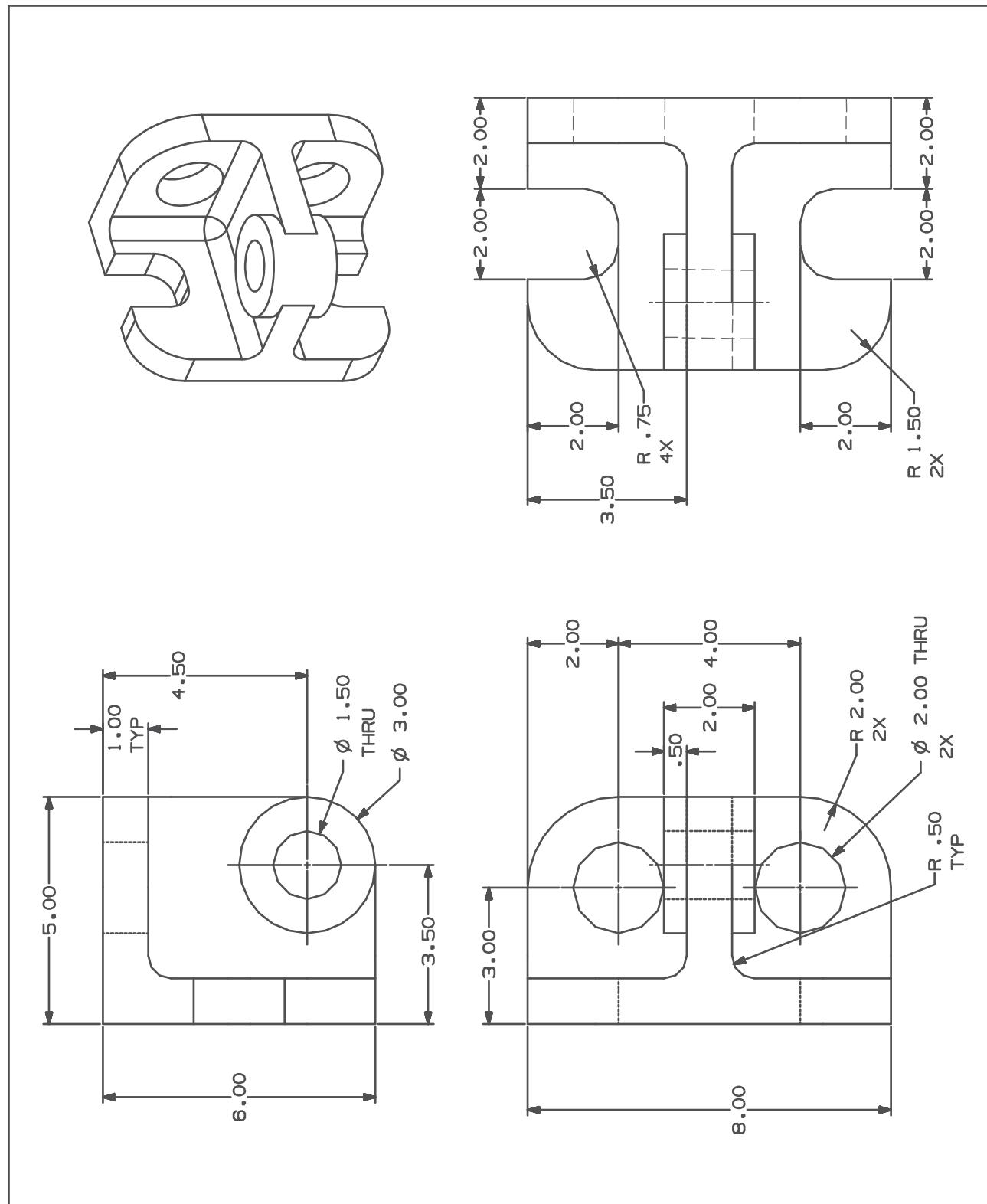


A

Practice Project 8

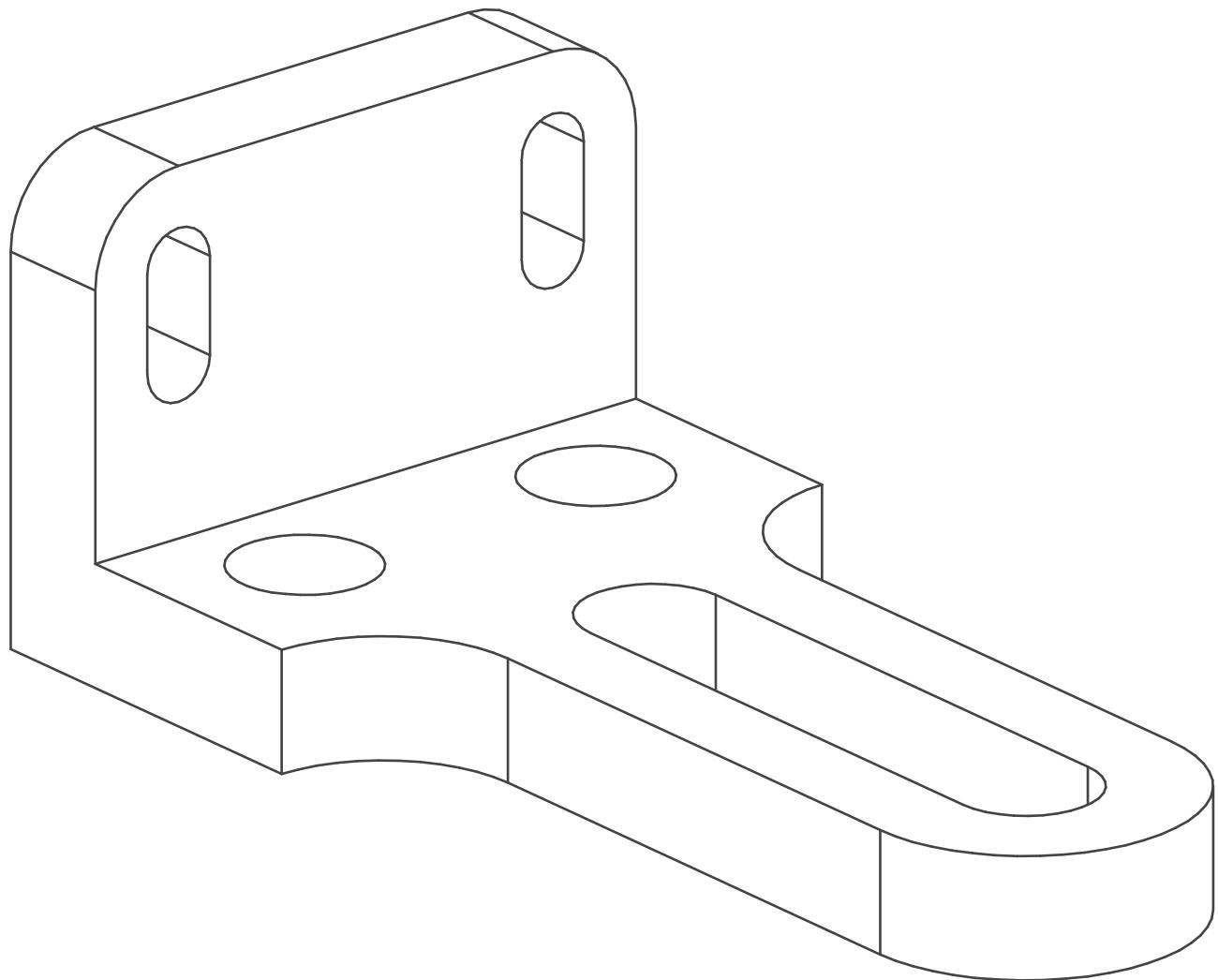


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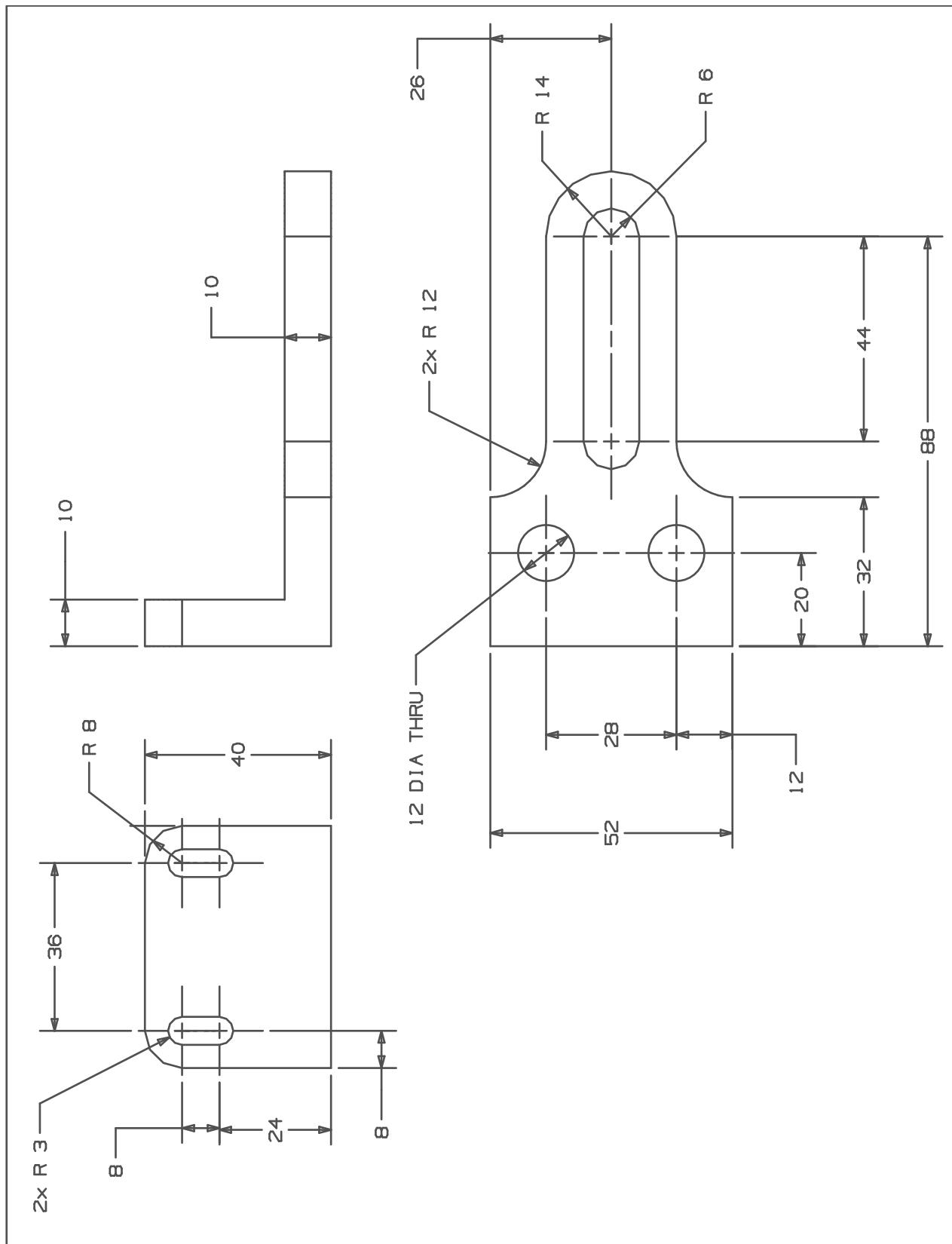


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Practice Project 9

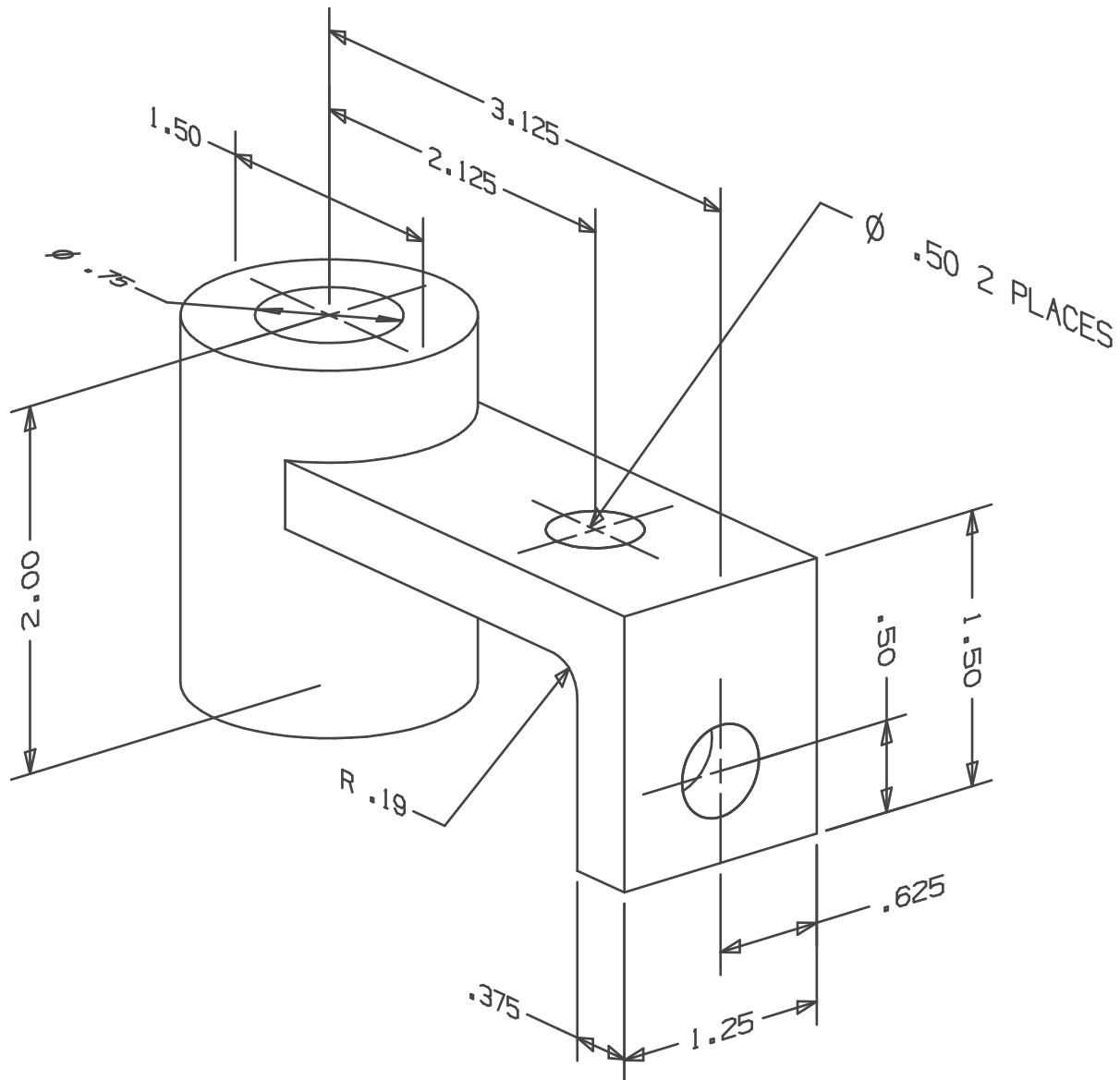


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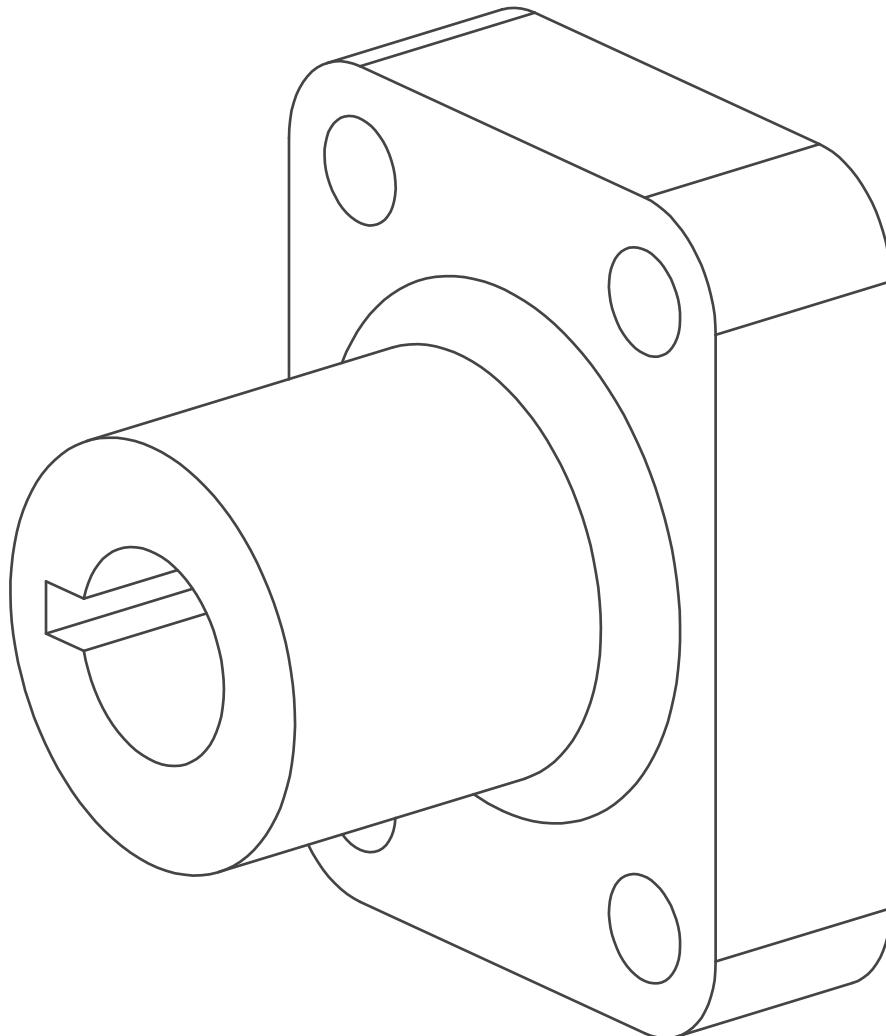
A

Practice Project 10

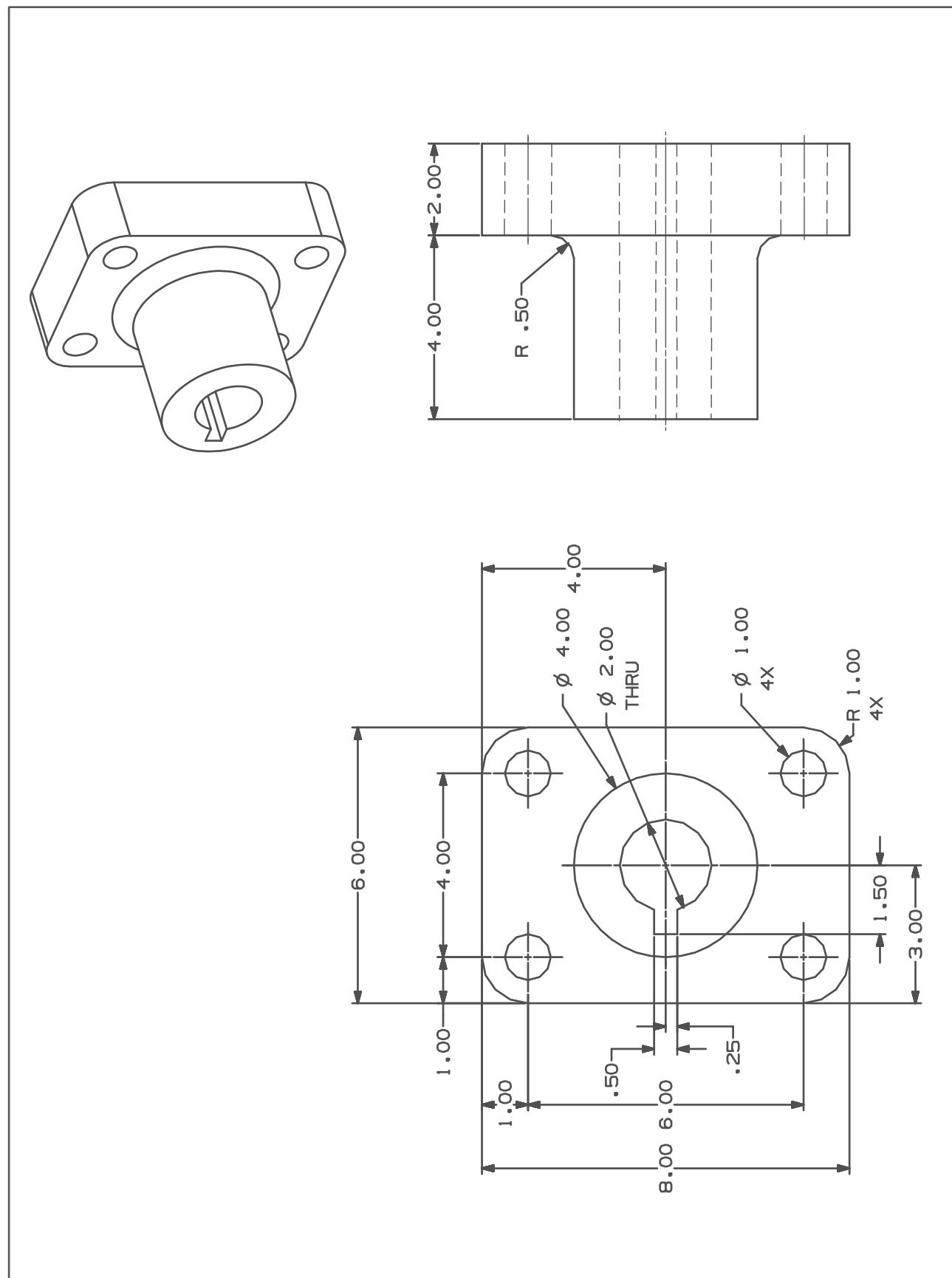


A

Practice Project 11

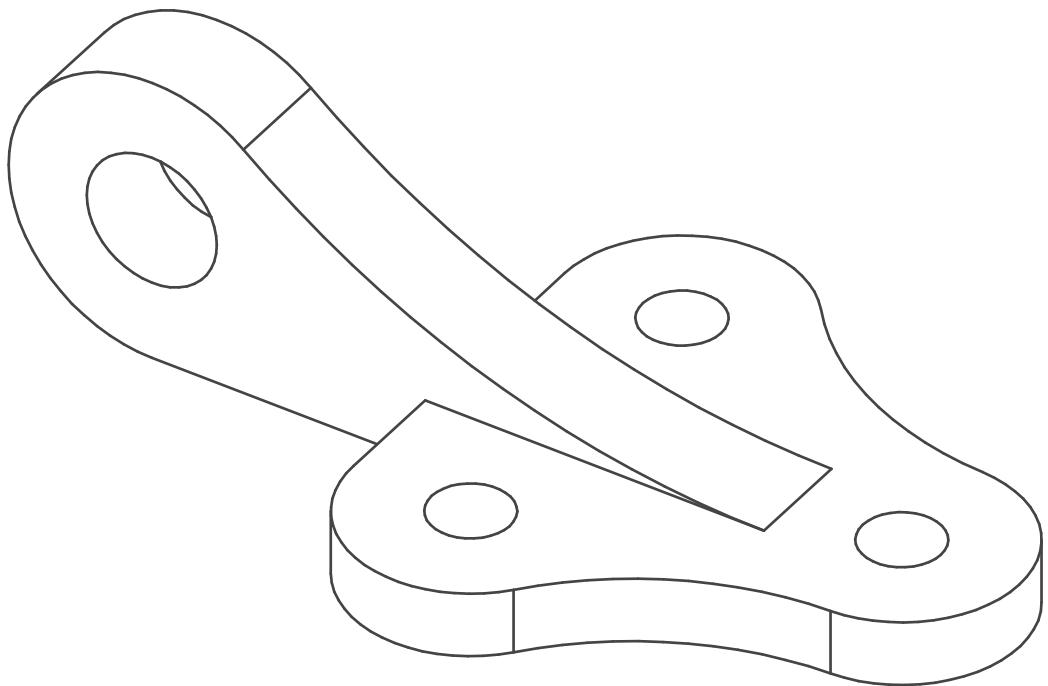


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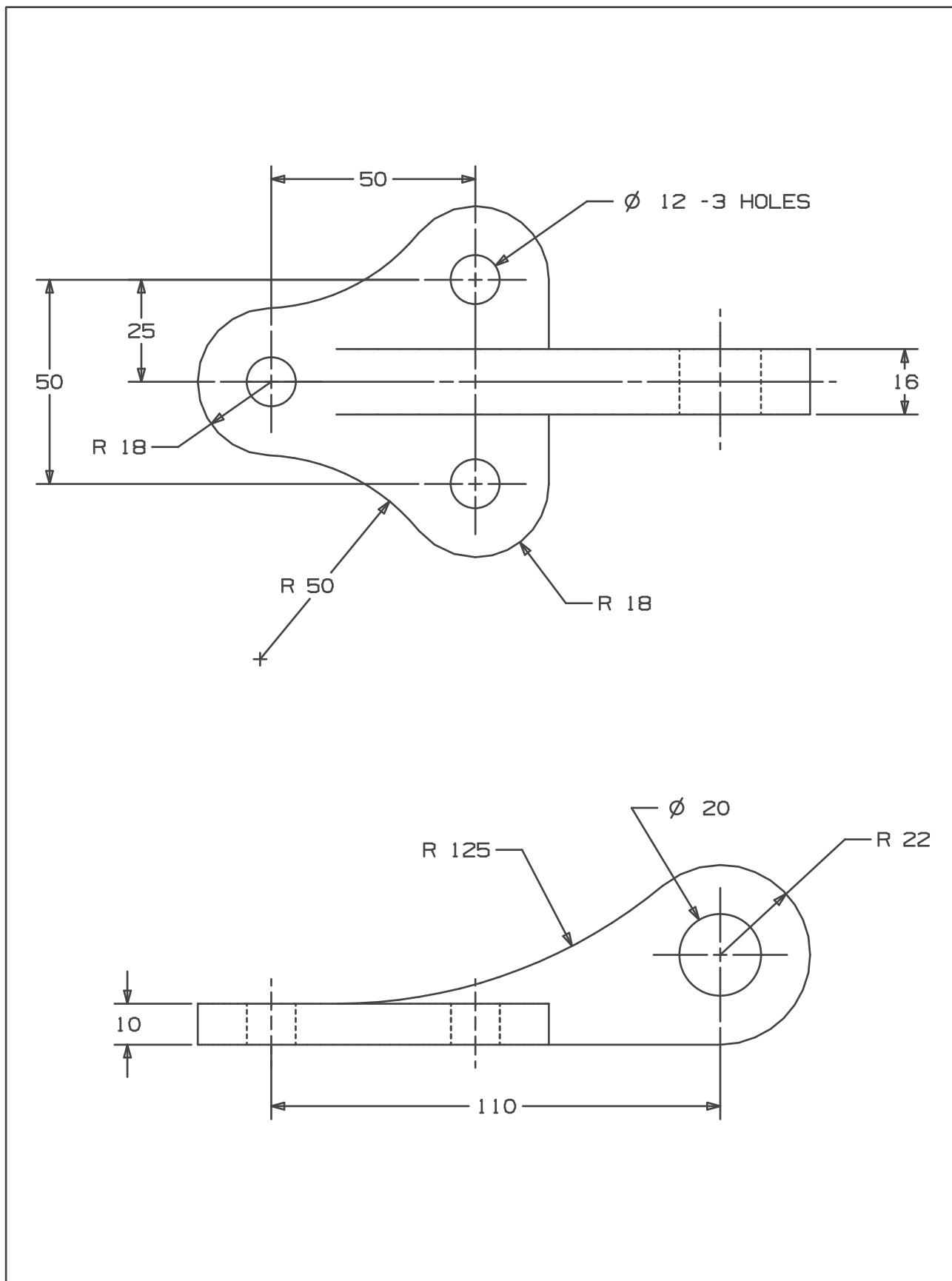


A

Practice Project 12

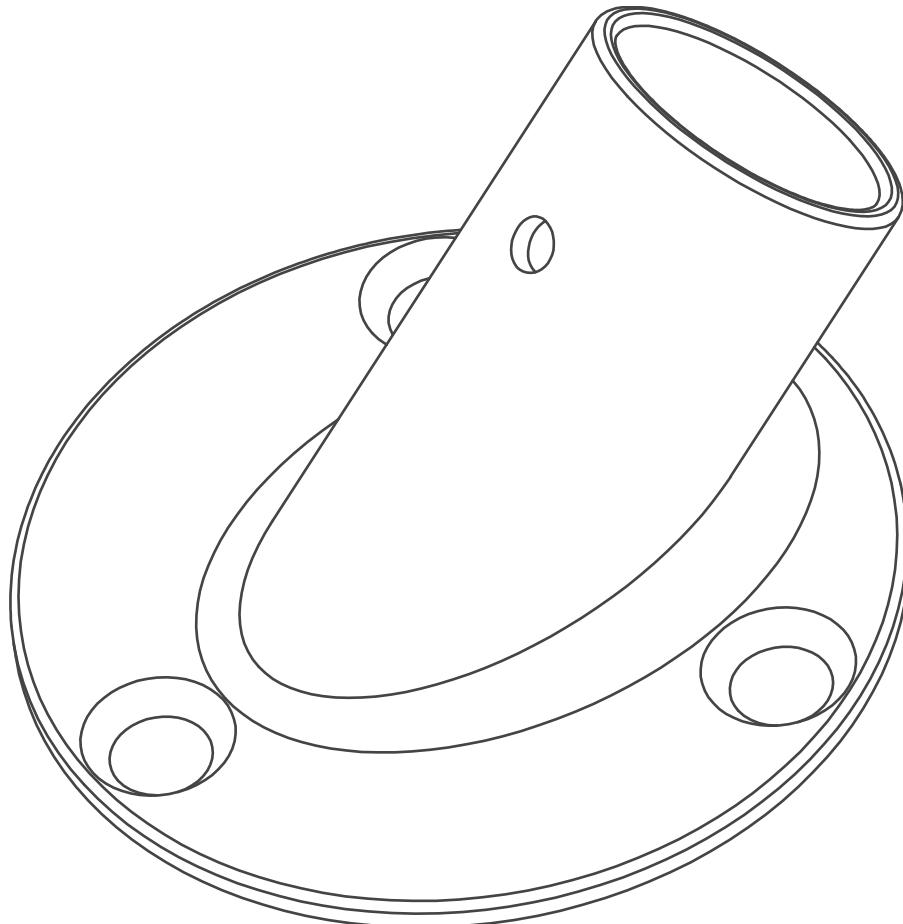


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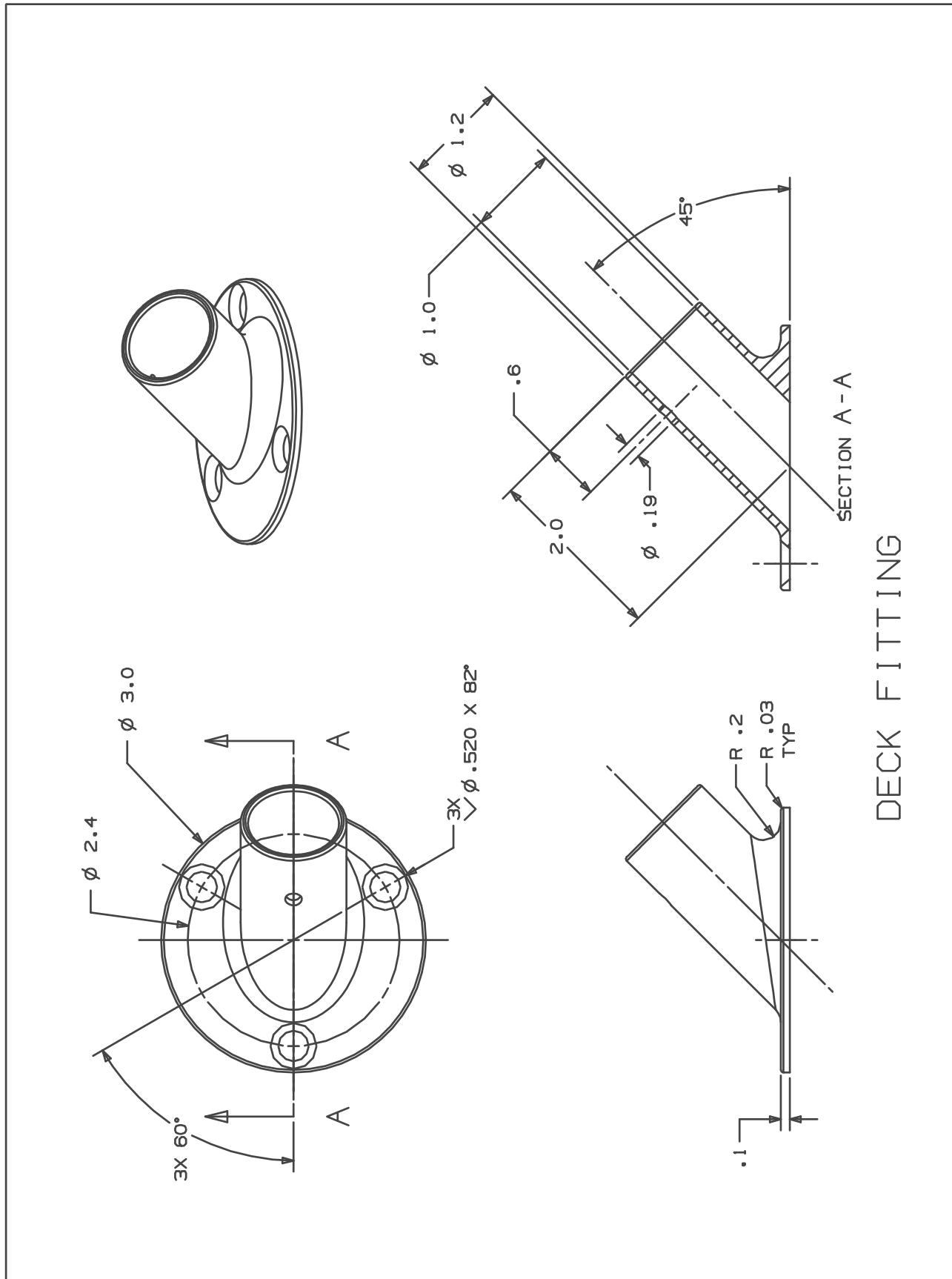


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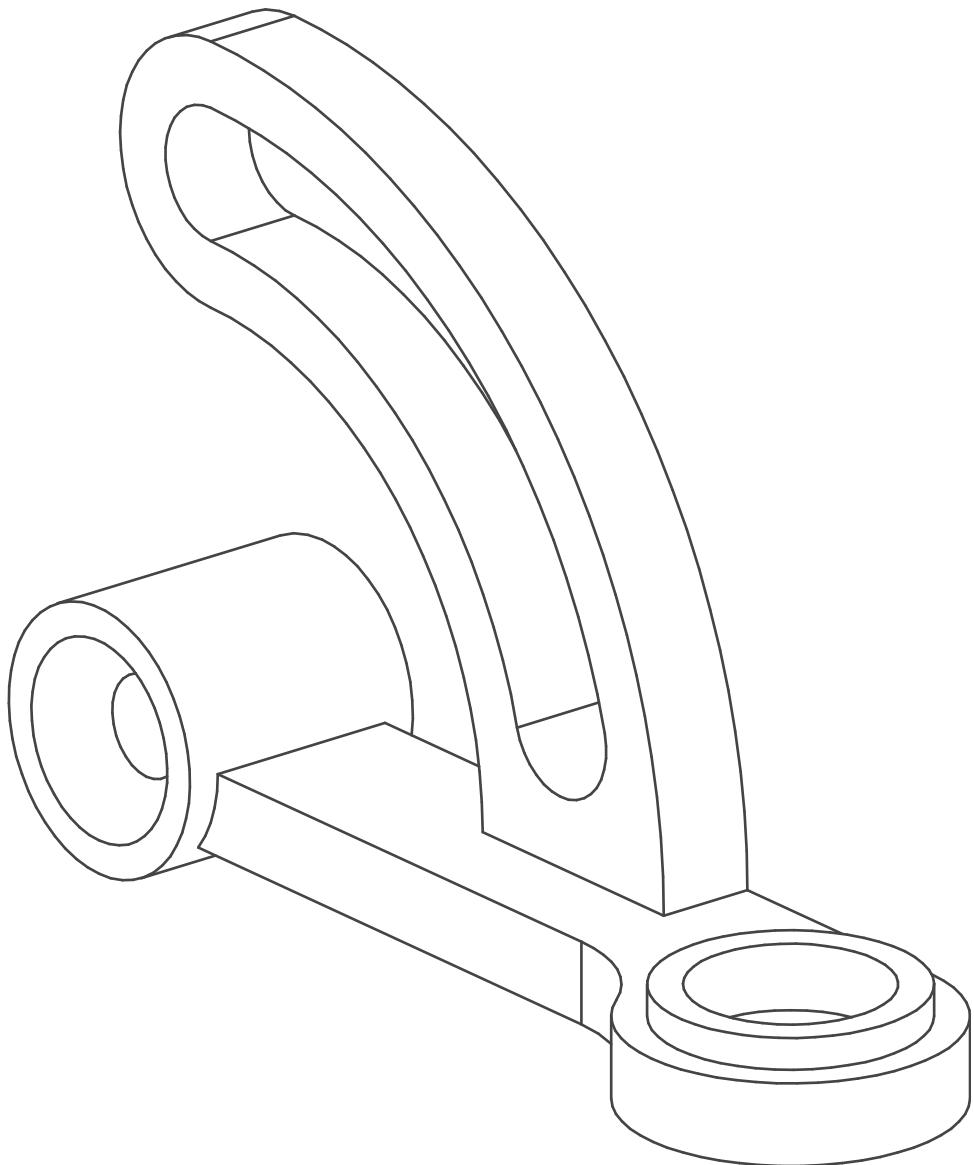
Practice Project 13



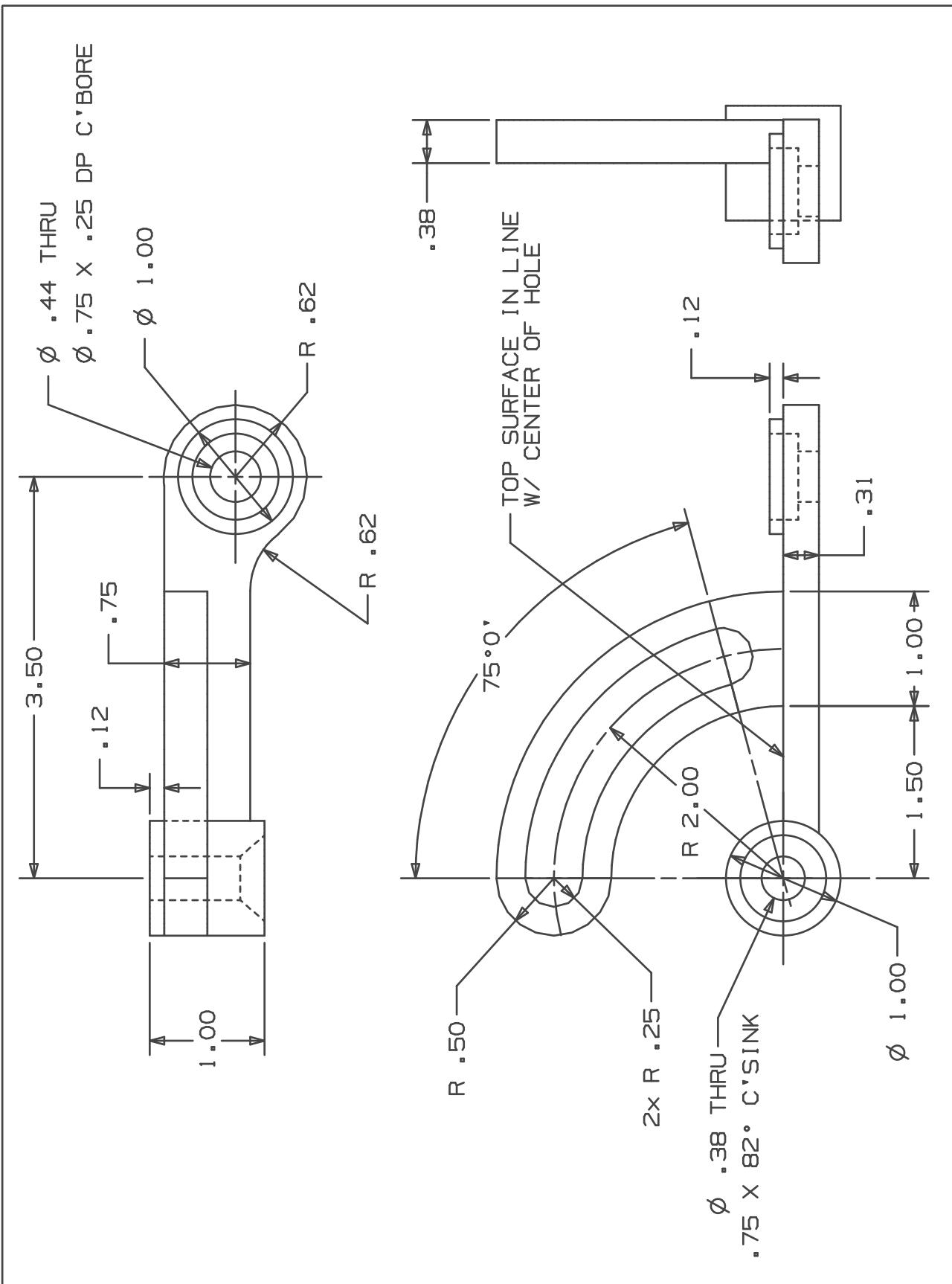
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**A**

Practice Project 14

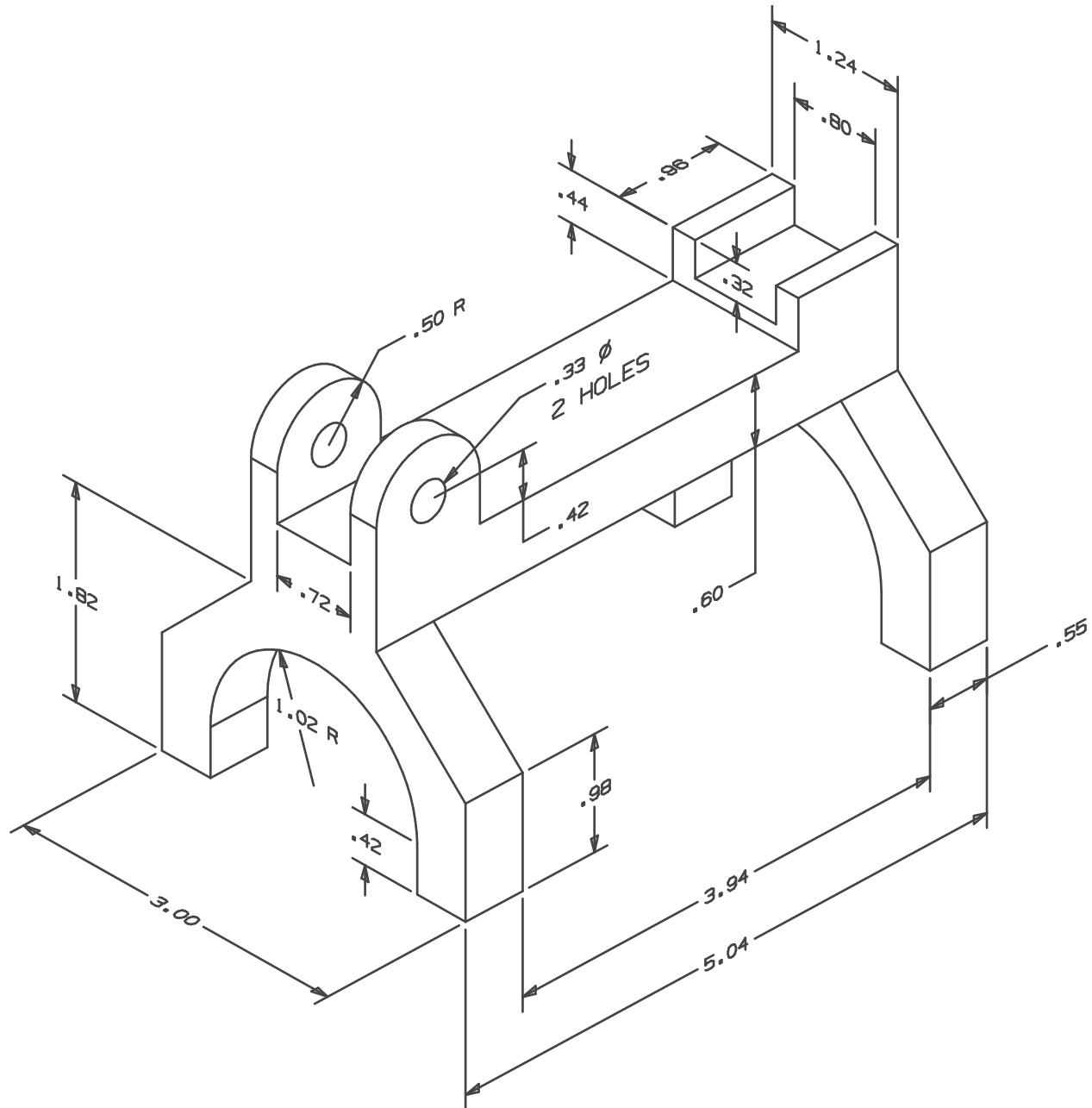


A



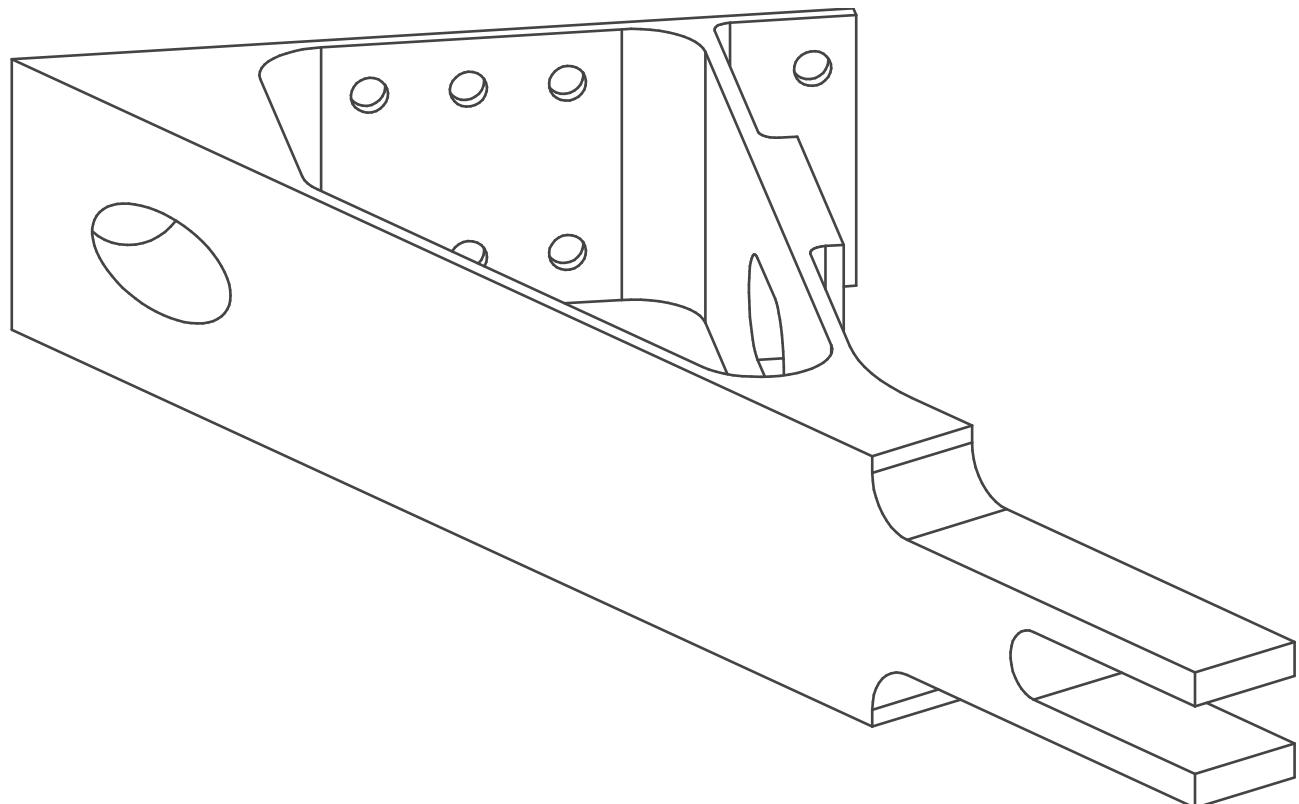
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Practice Project 15

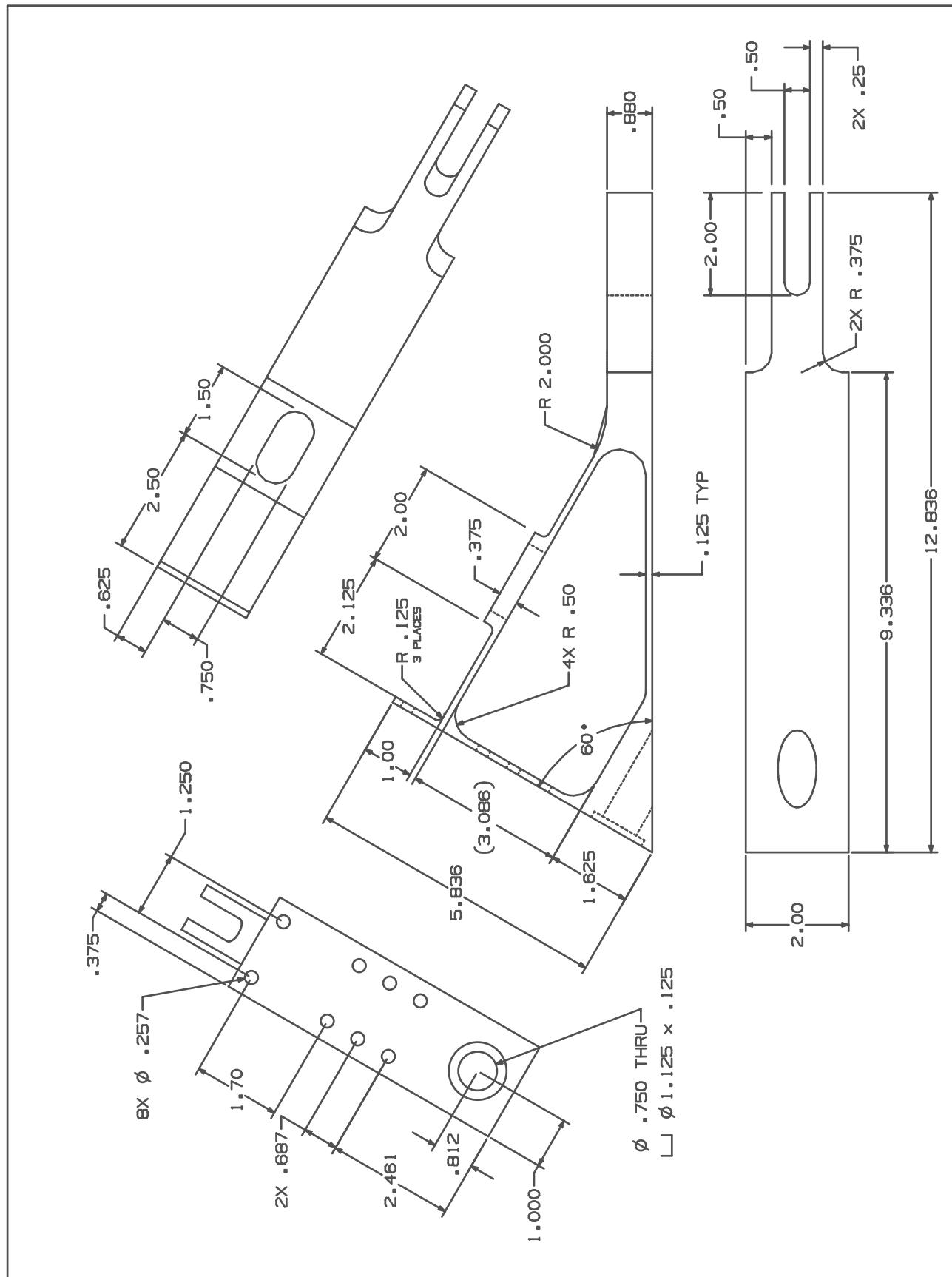


A

Practice Project 16

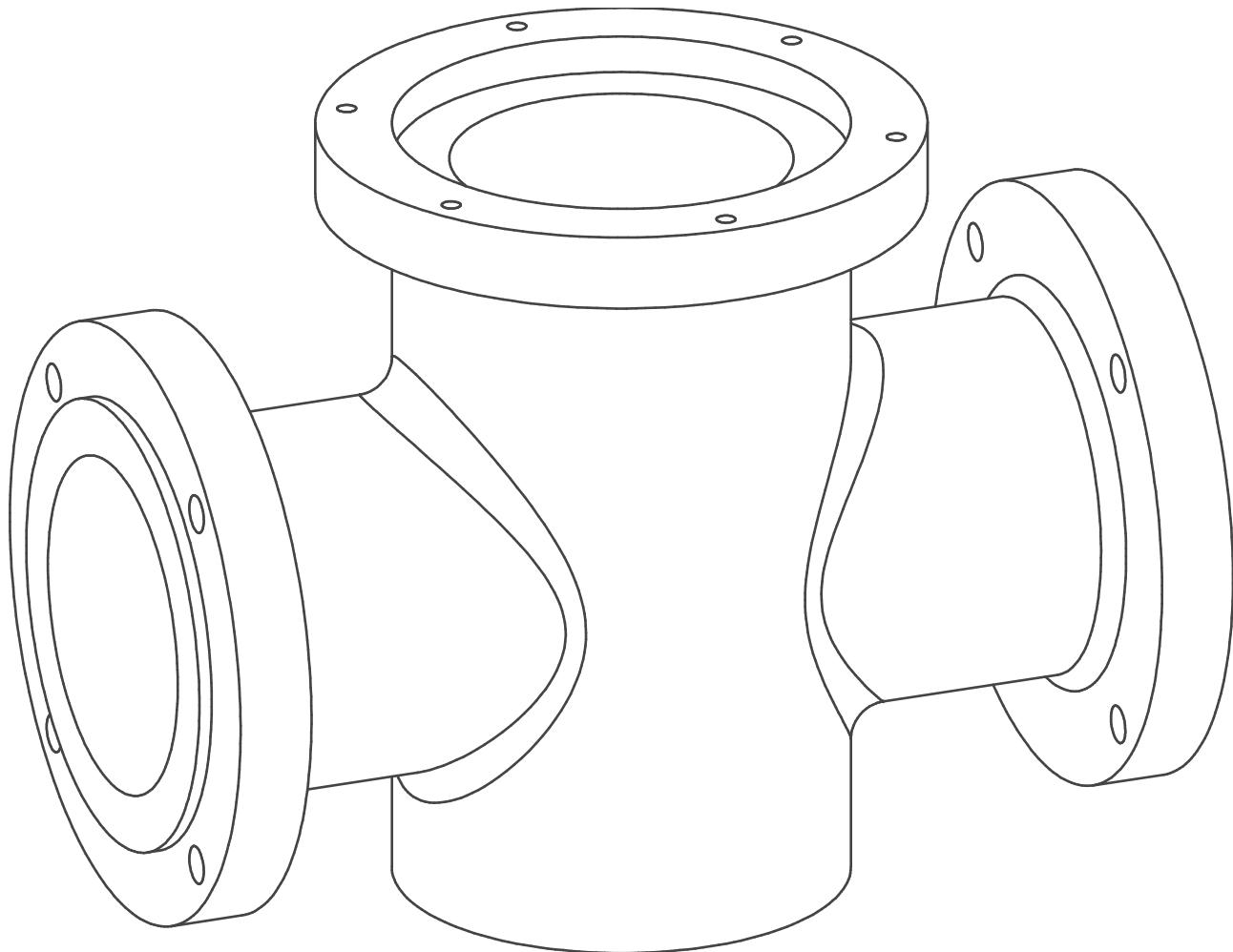


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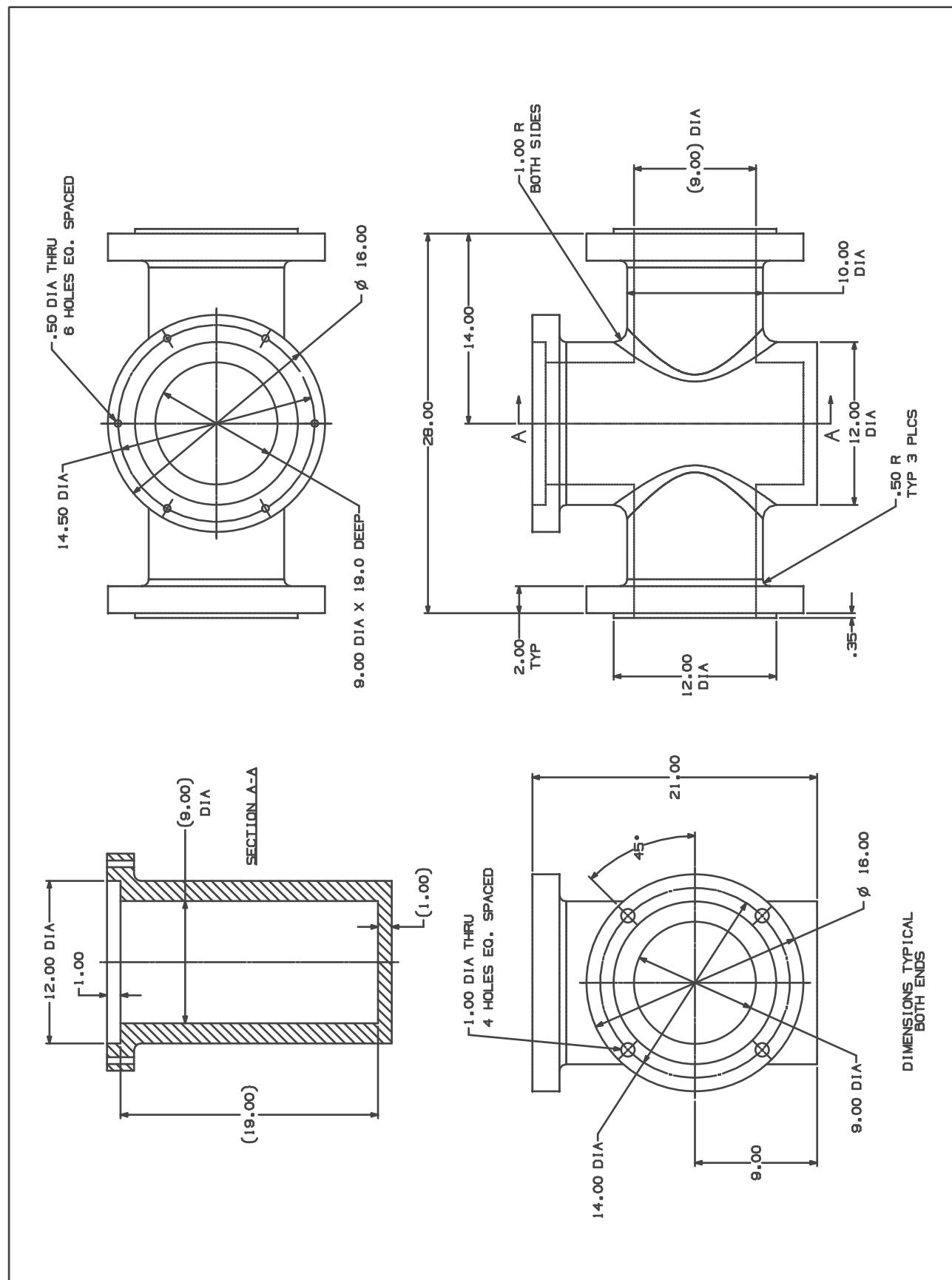


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Practice Project 17

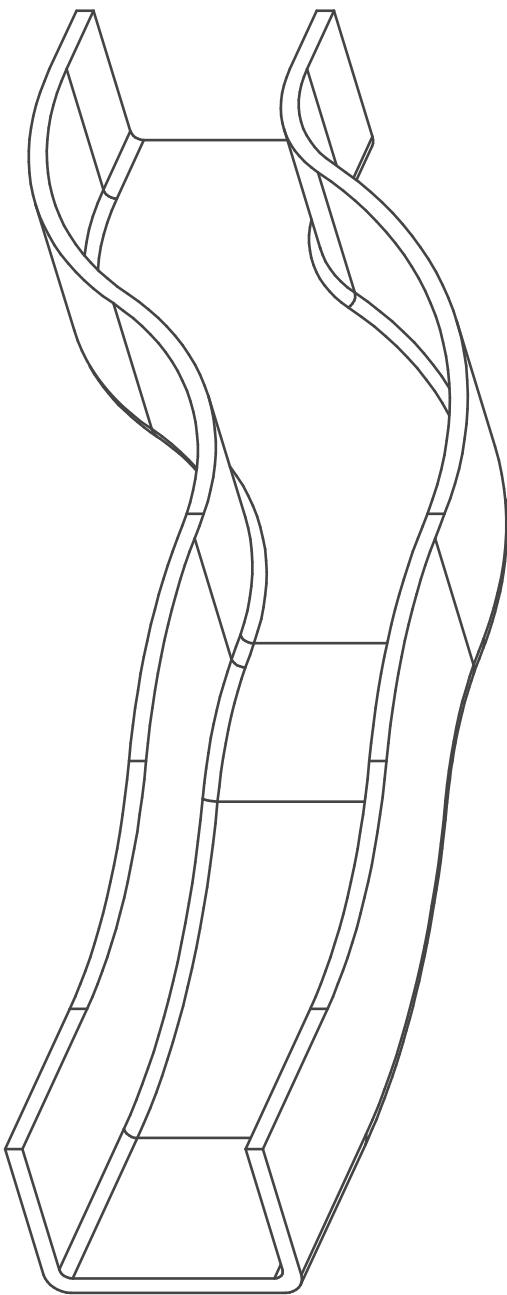


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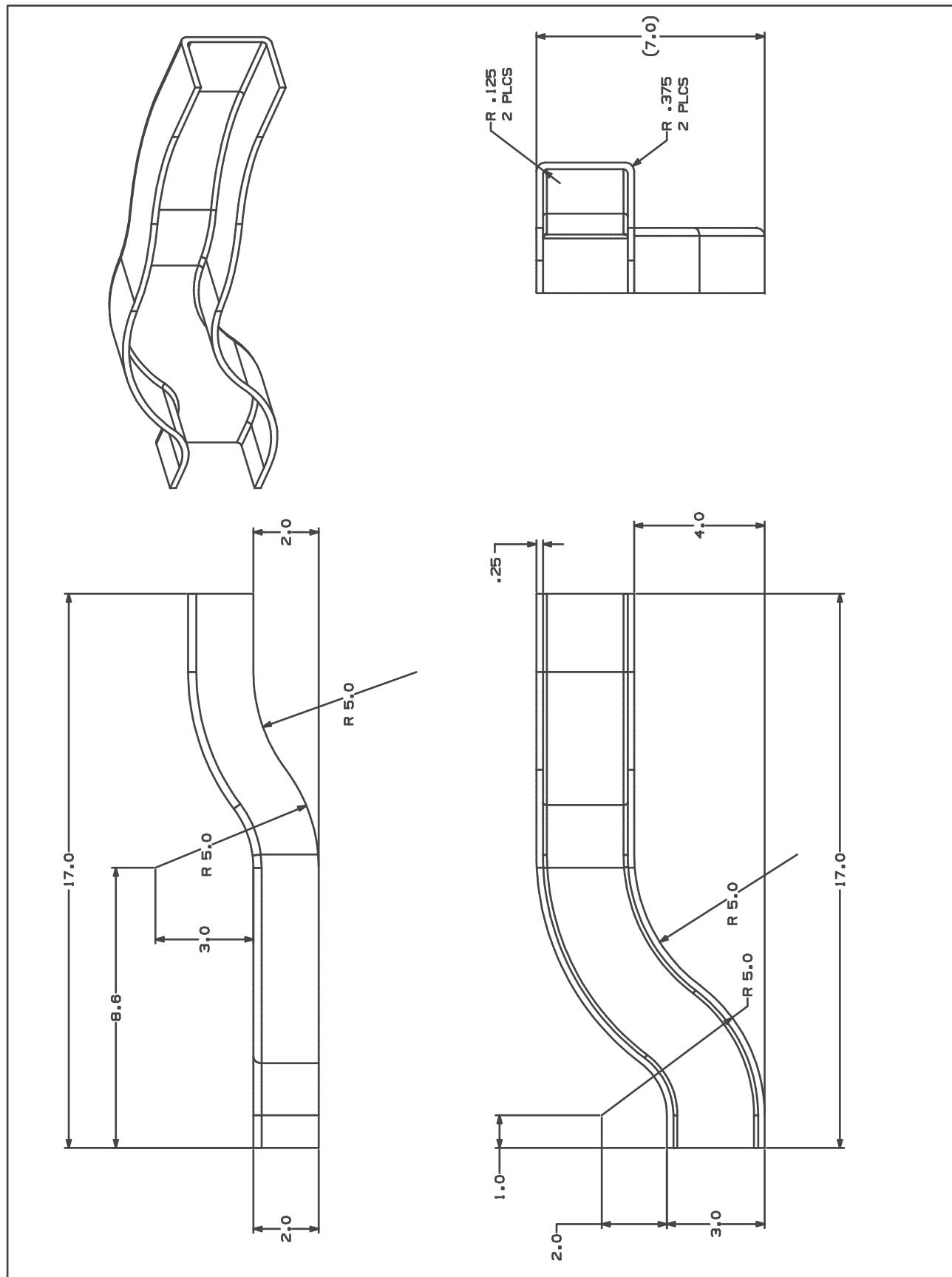


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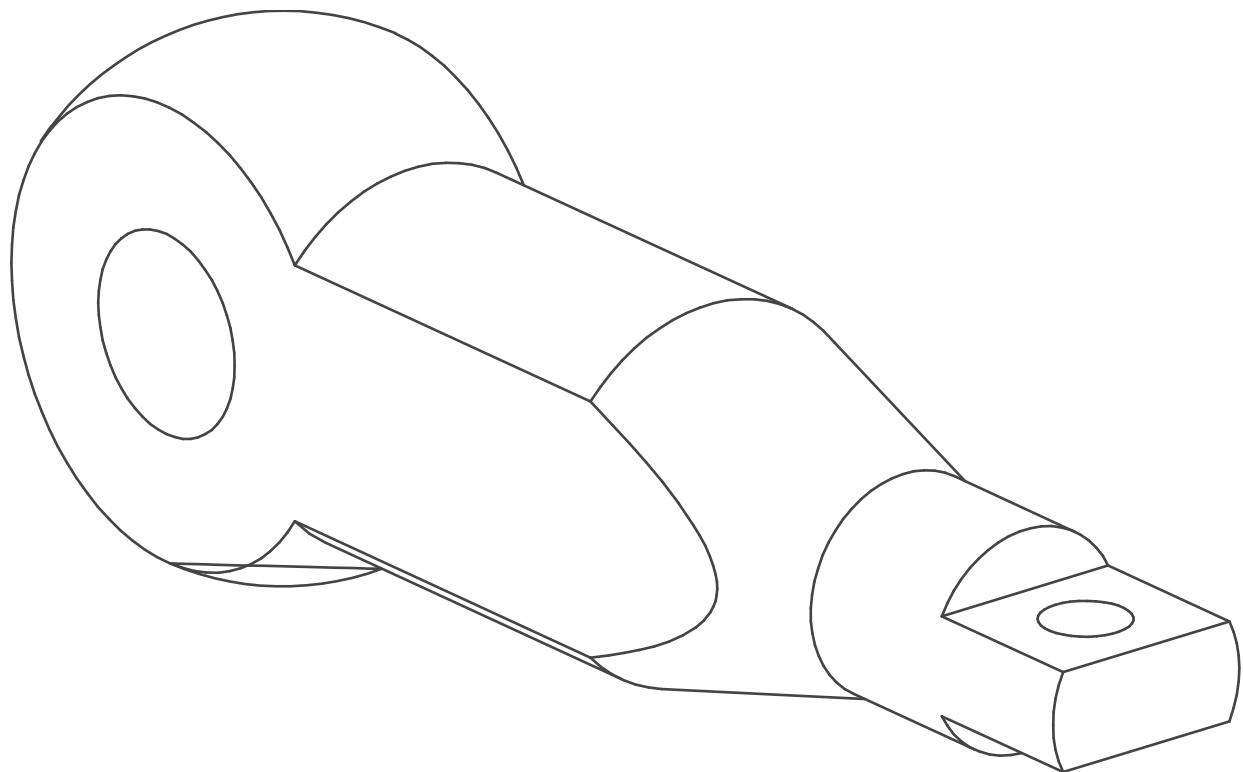
Practice Project 18



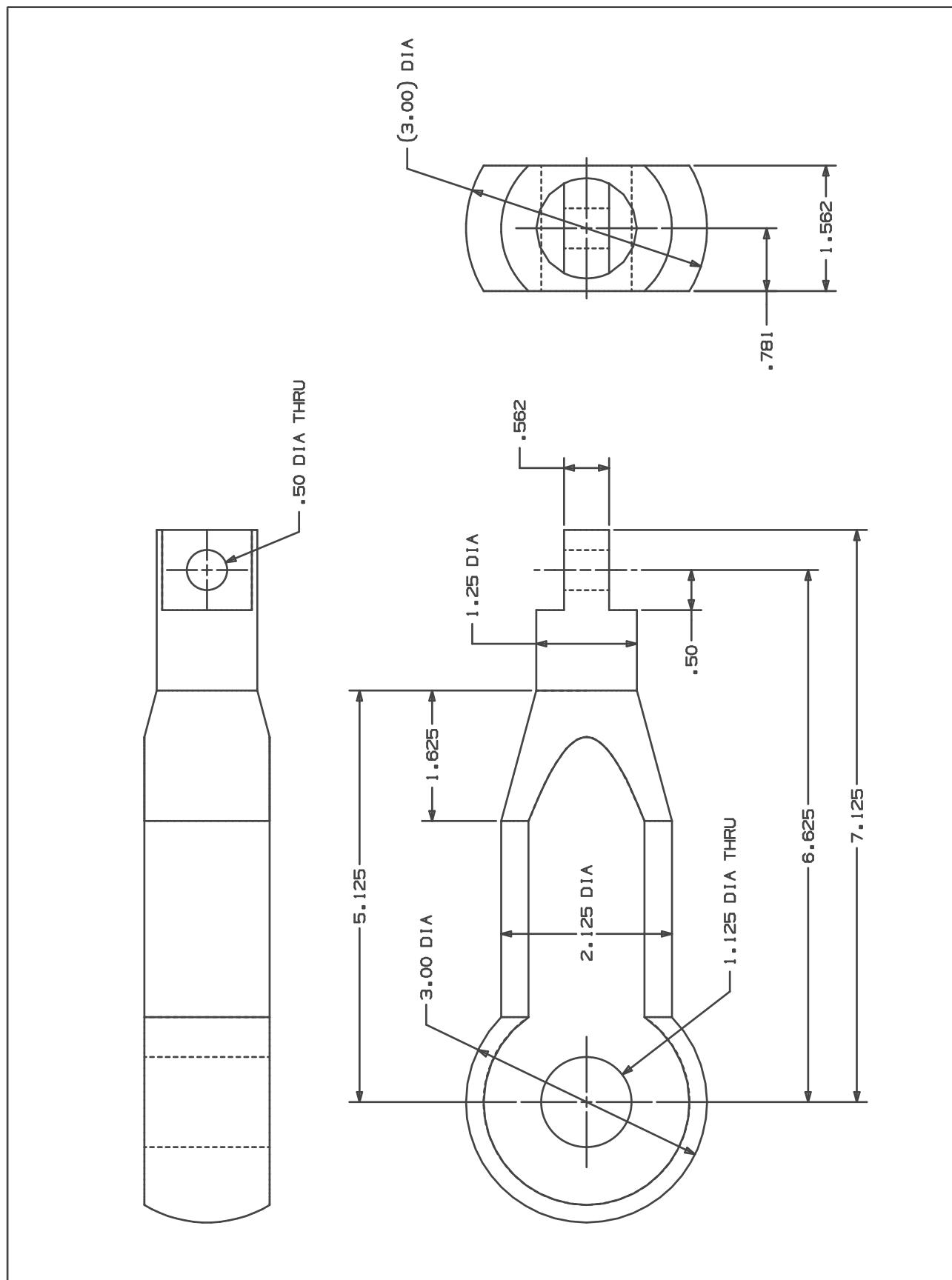
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Practice Project 19

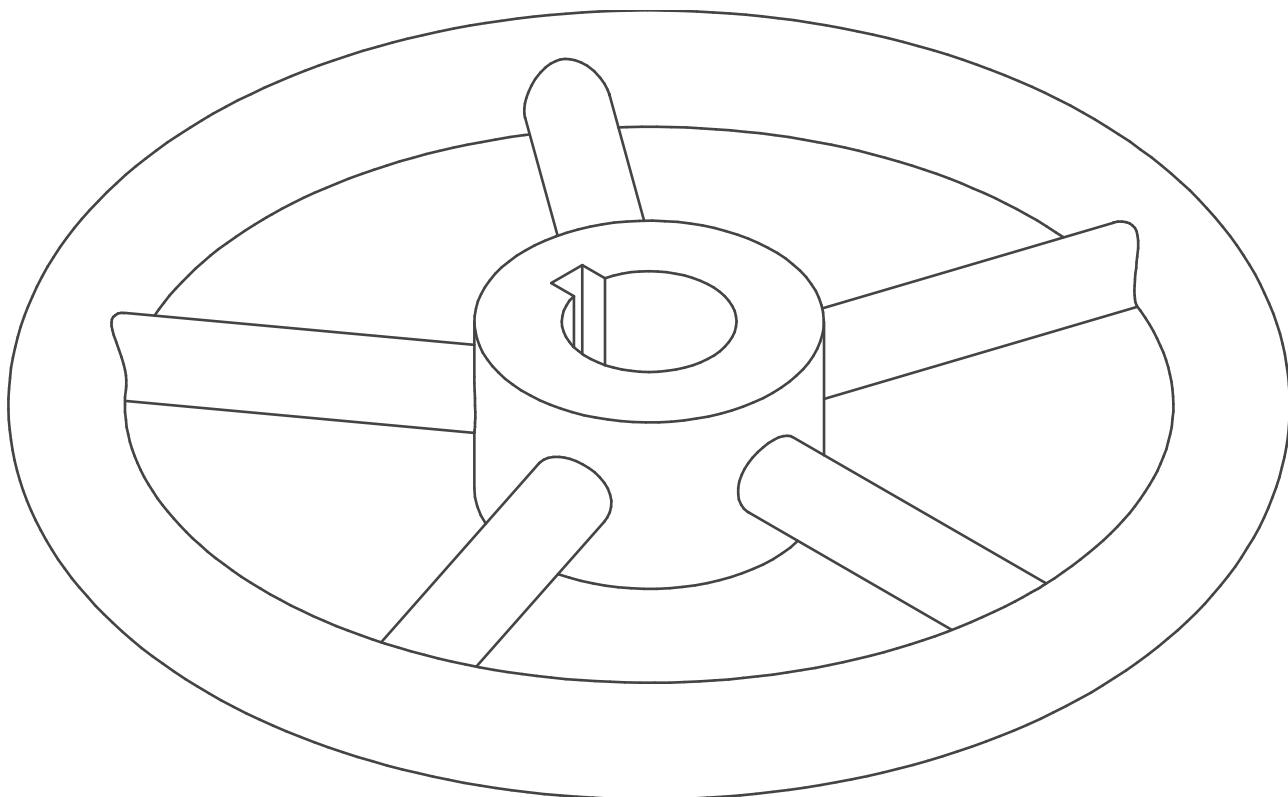


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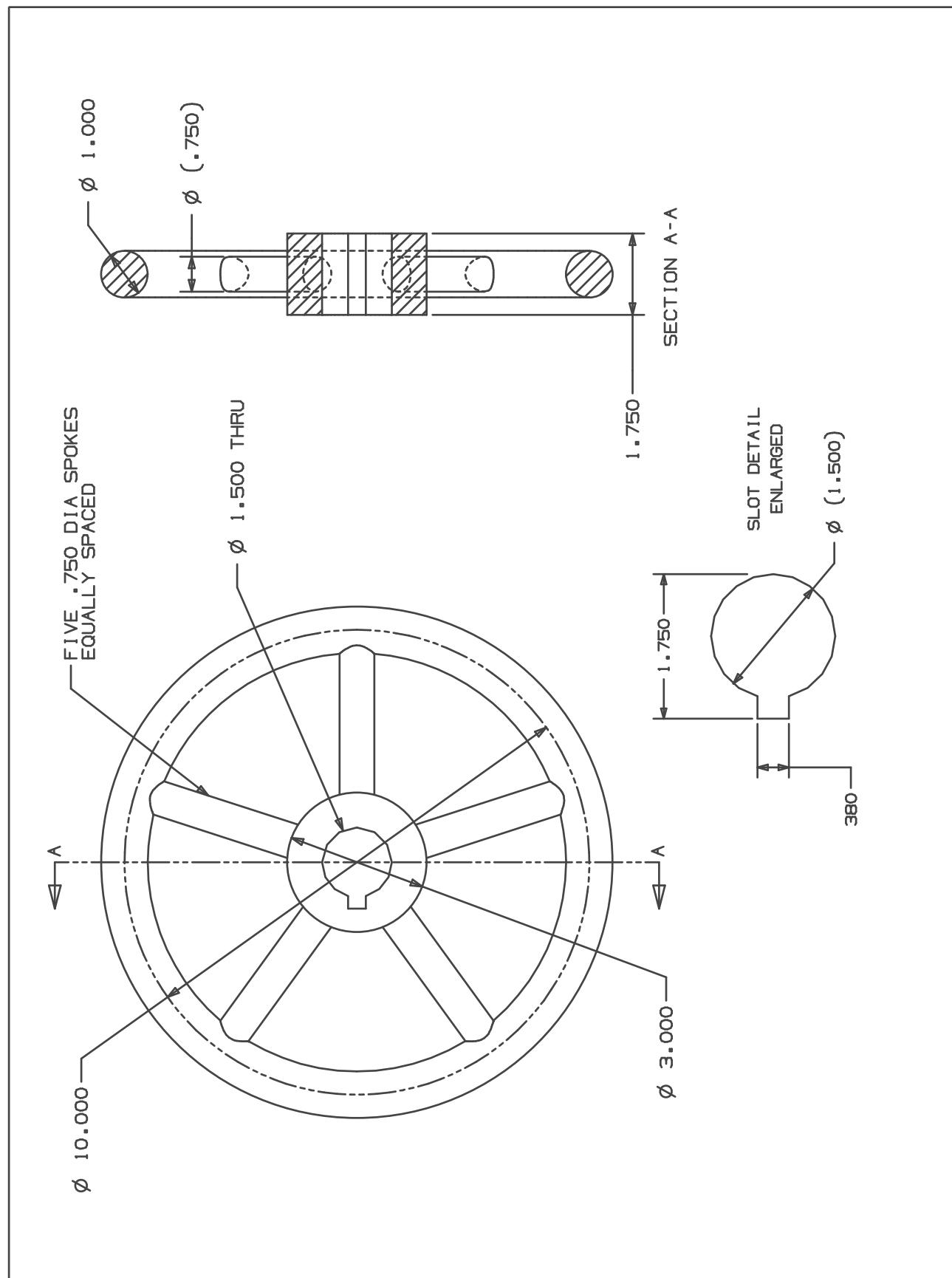


A

Practice Project 20

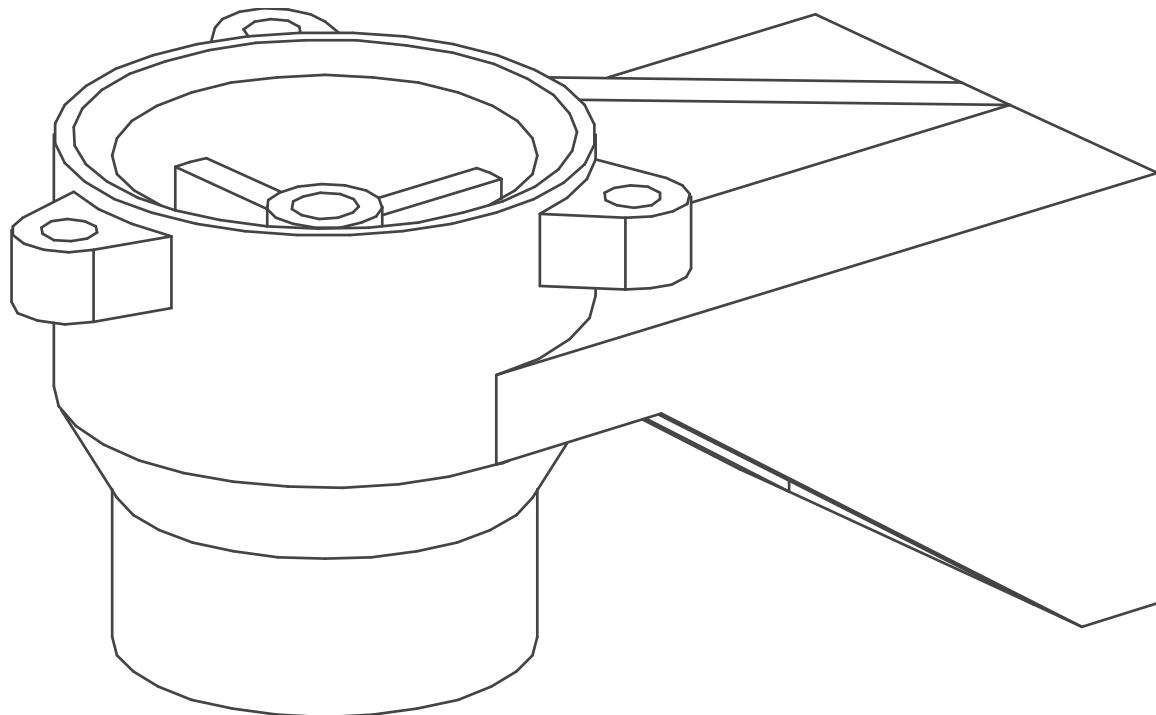


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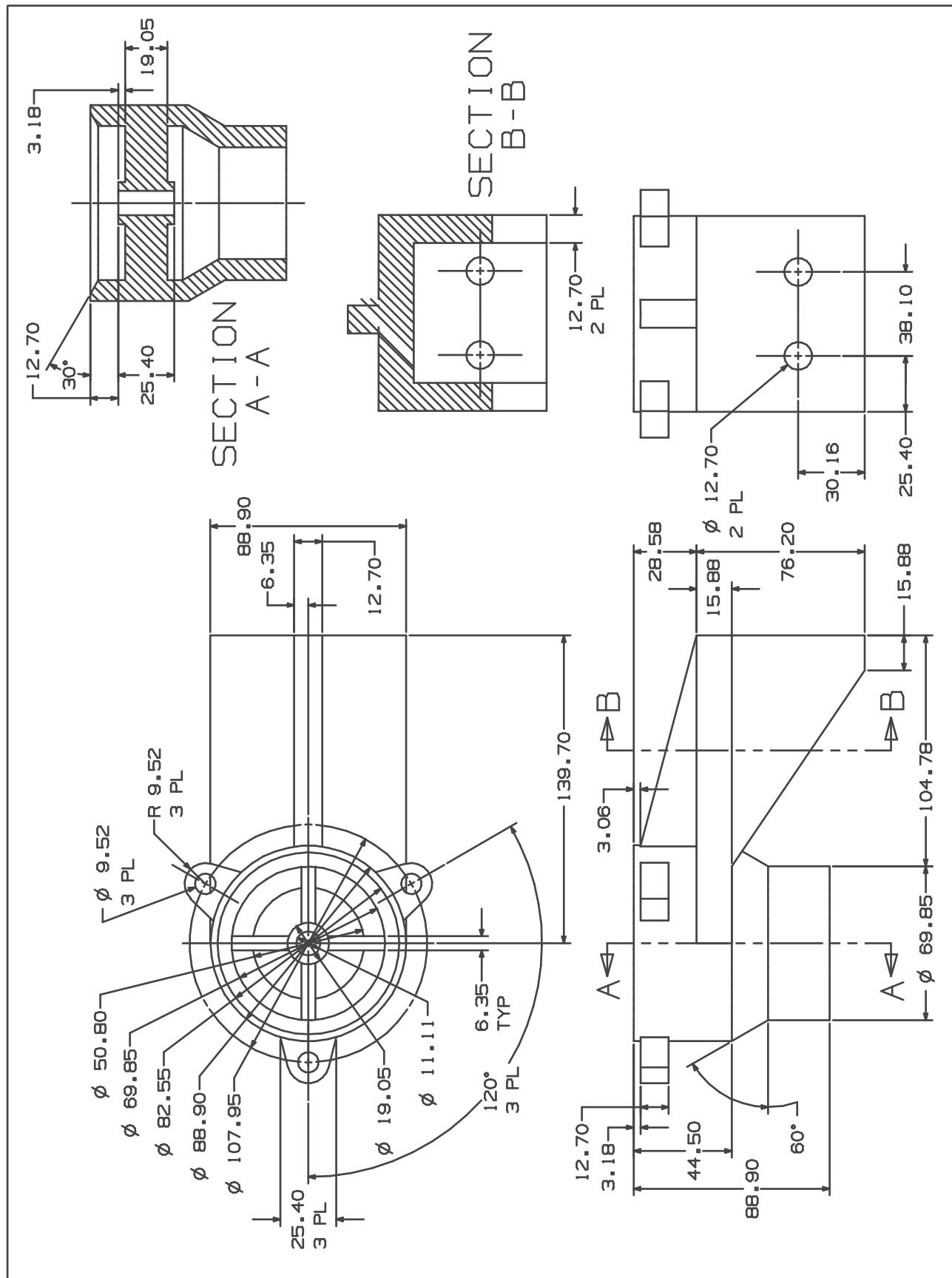


A

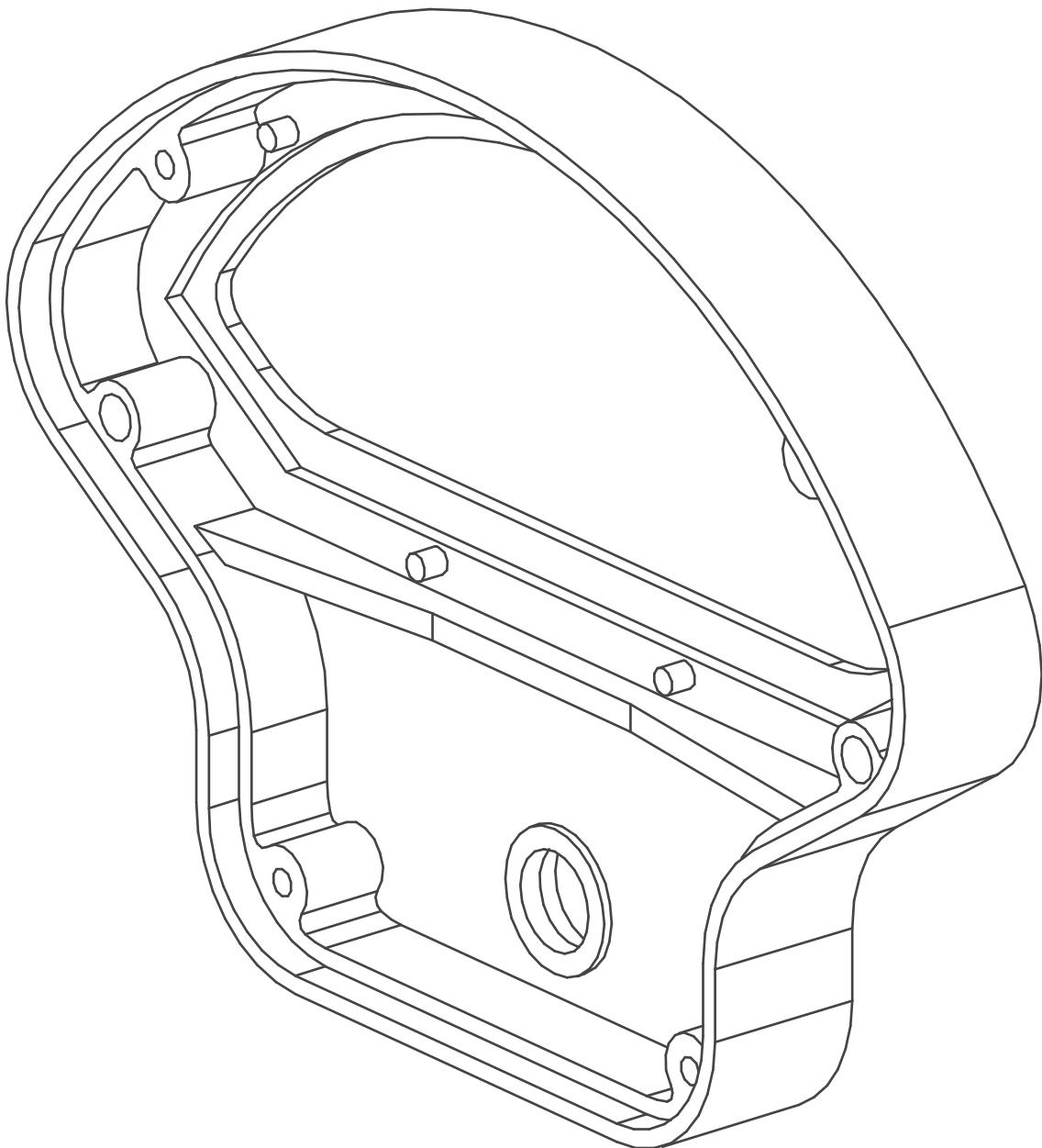
Practice Project 21



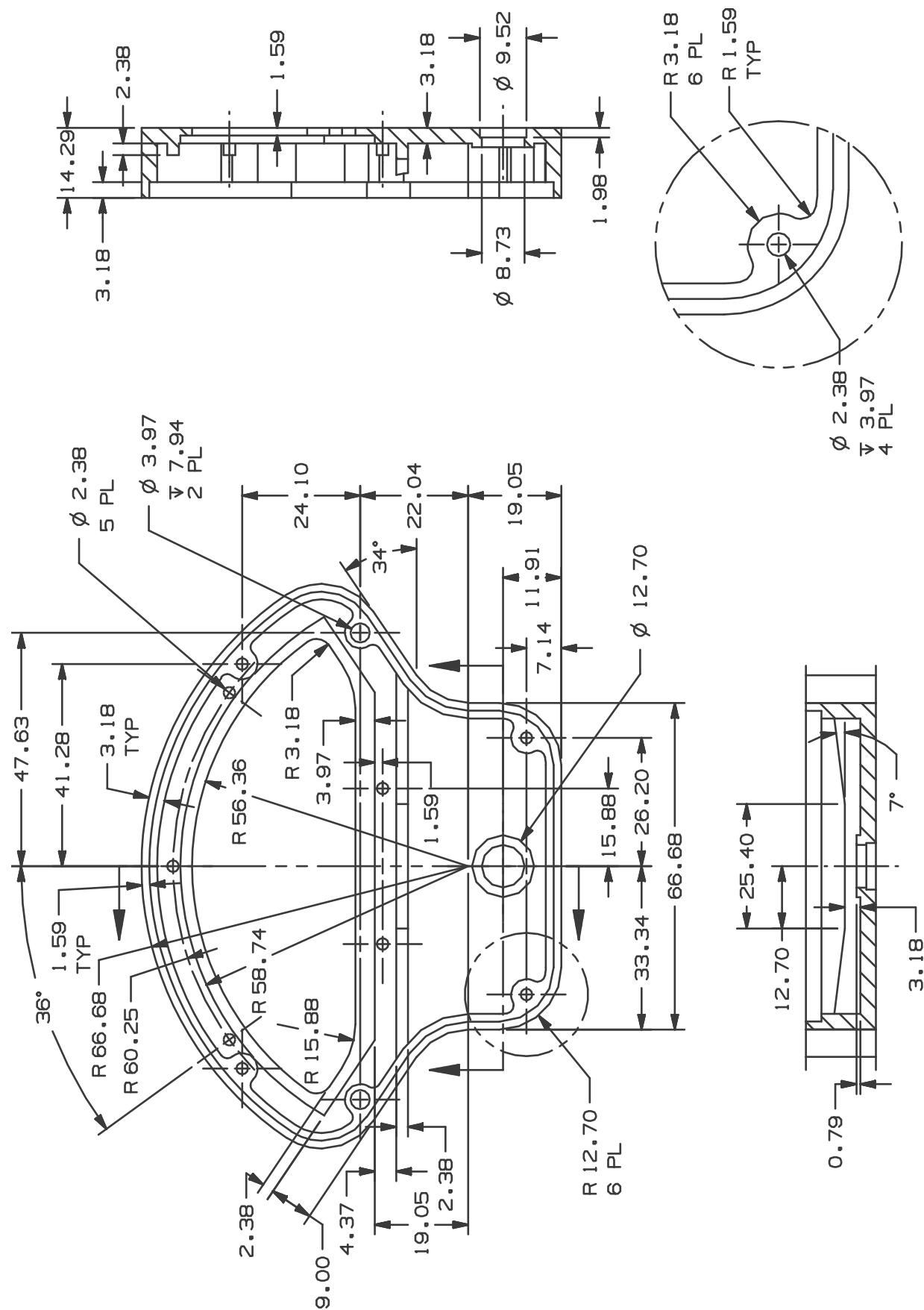
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Practice Project 22



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Appendix

B Expression operators

This appendix describes the operators and functions that you can use in expressions.

Operators

There are several types of operators that you may use in the expression language.

Arithmetic operators		Example
+	Addition	p2=p5+p3
-	Subtraction and Negative Sign	p2=p5-p3
*	Multiplication	p2=p5*p3
/	Division	p2=p5/p3
%	Modulus	p2=p5%p3
^	Exponential	p2=p5^2
=	Assignment	p2=p5

Relational and Boolean operators	
> Greater Than	
< Less Than	
>= Greater Than or Equal	
<= Less Than or Equal	
== Equal	
!= Not Equal	
! Negate	
& or && Logical AND	
or Logical OR	

Precedence and associativity

In the table below, operators in the same row have equal precedence while operators in the following rows have less precedence.

Precedence and associativity	
Operators	Associativity
<code>^</code>	Right to left
<code>-</code> (change sign)	
<code>*</code> <code>/</code> <code>%</code>	Left to right
<code>+</code> <code>-</code>	
<code>></code> <code><</code> <code>>=</code> <code><=</code>	
<code>==</code> <code>!=</code>	
<code>&&</code>	
<code> </code>	
<code>=</code>	Right to left

When using operators with the same precedence in an equation without parameters, use left-to-right or the right-to-left rule from the table. For example:

$$X = 90 - 10 + 30 = 110 \text{ (not 50)}$$

$$X = 90 - (10 + 30) = 50$$

Legacy unit conversion

Although when dimensionality is specified and units are assigned the system handles conversions, legacy parts may have used functions for unit conversion. For legacy compatibility these functions are supported.

Functions for unit conversion

cm	cm(x) converts x from centimeters to the default units of the part
ft	ft(x) converts x from feet to the default units of the part
grd	grd(x) converts x from gradients to degrees
in	in(x) converts x from inches to the default units of the part
km	km(x) converts x from kilometers to the default units of the part
mc	mc(x) converts x from microns to the default units of the part
min	min(x) converts x from minutes to degrees.
ml	ml(x) converts x from mils to the default units of the part
mm	mm(x) converts x from millimeters to the default units of the part
mtr	mtr(x) converts x from meters to the default units of the part
sec	sec(x) converts x from seconds to degrees
yd	yd(x) converts x from yards to the default units of the part

Built-in functions

Built-in functions include math, string, and engineering functions.

Scientific notation

You may optionally enter numbers in *scientific notation*. The value you enter must contain a positive or negative sign. For example, you can enter:

$2e+5$ which is the same as the value 200000

$2e-5$ which is the same as the value .00002

Built-in functions	
abs	Returns the absolute value of a given number
arccos	Returns the inverse cosine of a given number in degrees
arcsin	Returns the inverse sine of a given number in degrees
arctan	Returns the inverse tangent of a given number in degrees from -90 to +90
arctan2	Returns the inverse tangent of a given delta x divided by a given delta y in degrees from -180 to +180
ASCII	Returns the ASCII code of the first character in a given string or zero if the string is empty
ceiling	Returns the smallest integer that is bigger than a given number
Char	Returns the ASCII character for a given integer in the range 1 to 255
charReplace	Returns a new string from a given source string, character to replace and the corresponding replacement characters.
compareString	Case sensitive compare of two strings
cos	Returns the cosine of a given number in degrees
dateTimeString	Returns the system date and time in the format "Fri Nov 21 09:56:12 2005/n"
floor	Returns the largest integer less than or equal to a given number
format	Returns a formatted string, using C-style formatting specification
getenv	Returns the string value of a given environment variable
hypcos	Returns the hyperbolic cosine of a given number
hypsin	Returns the hyperbolic sine of a given number
hyptan	Returns the hyperbolic tangent of a given number

Built-in functions	
log	Returns the natural logarithm of a given number
log10	Returns the logarithm base 10 of a given number
MakeNumber	Returns the number or integer of a given numerical string
max	Returns the largest number from a given number and additional numbers
min	Returns the smallest number from a given number and additional numbers
mod	Returns the remainder (modulus) when a given numerator is divided by a given denominator (by integer division)
NormalizeAngle	Normalizes a given angle (degrees) to be between 0 and 360 degrees
pi()	Returns pi
Radians	Converts an angle in degrees into radians
replaceString	Replaces all occurrences of str1 with str2
round	Returns the integer nearest to a given number, returns the even integer if the given number ends in .5
sin	Returns the sine of a given number in degrees
sqrt	Returns the inverse square root of a given positive number
StringLower	Returns a lowercase string from a given string
StringUpper	Returns an uppercase string from a given string
StringValue	Returns a string containing a textual representation of a given value
subString	Returns a new string containing a subset of the elements from the original list
tan	Returns the sine of a given number
ug_ functions	see the documentation for descriptions of dozens more specialized math and engineering functions

Appendix

C *System Topics overview*

There are utilities and files which affect the interface and behavior of the system.

This appendix covers these topics which would normally be the responsibility of a system administrator.

Customer Defaults

Customer defaults are accessed by choosing **File→Utilities→Customer Defaults.**

When NX is first started (out-of-the-box) the defaults are set to *User* and a variable points to a user file which may or may not exist. This is an extract from the log file for a user named “nxuser” after logging in and starting NX for the first time:

```
Processing customer default values file
C:/Documents and Settings/nxuser
/Local Settings/Application Data/Unigraphics Solutions
/NX75/nx75_user.dpv

User customizations file
C:/Documents and Settings/nxuser
/Local Settings/Application Data/Unigraphics Solutions
/NX75/nx75_user.dpv does not exist
```

The fact that the file does not exist is of no concern because the path is writable for the person logged in.

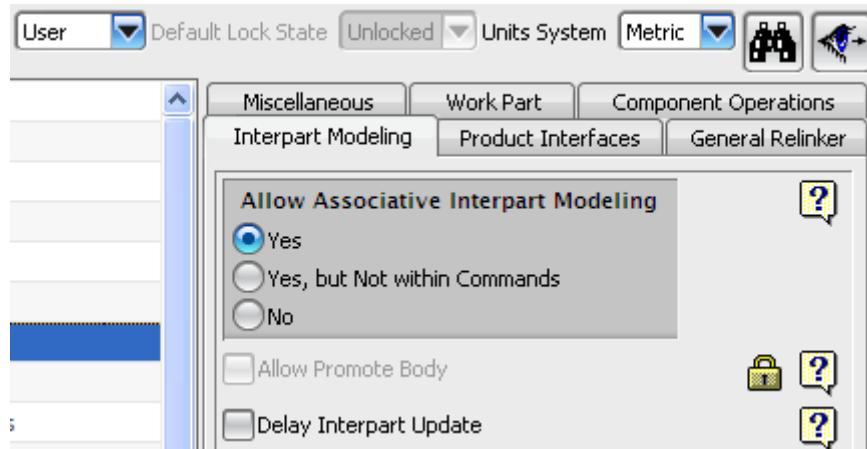
NX will create the file nx75_user.dpv when and if the user makes a change to the defaults.

If the administrator wishes to prevent the user from changing the defaults, i.e., set them as *User (Read Only)*, there are various ways to accomplish it:

- Create the file and customize it as you wish, and then make it read only.
- Define the file in a path to which the user cannot write. The file and the path need not exist.
- Lock one or more defaults at a higher level, i.e. group or site level.

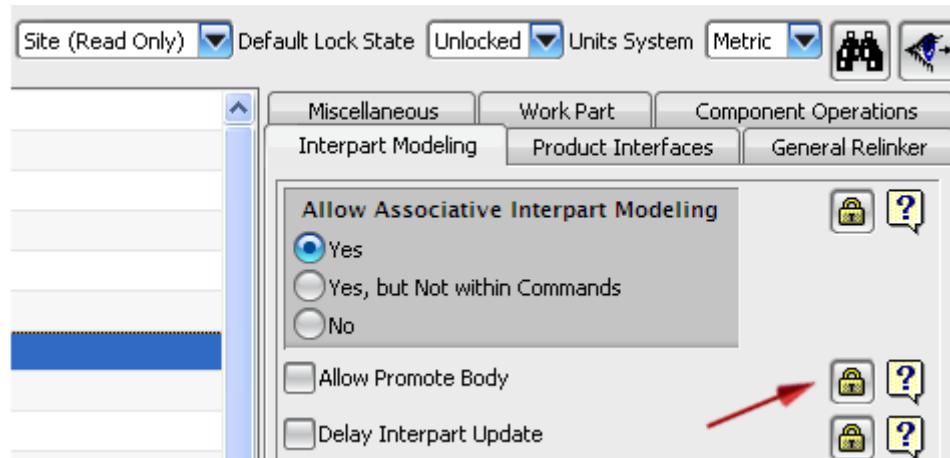
Customer Defaults levels

Customer defaults can be controlled at three levels: Site, Group, and User. Site is the highest level, User is the lowest. Any or all of these levels may be available to you, based on how the customer default environment variables are defined at your site. If none of the environment variables are defined, the level is Shipped (read-only).

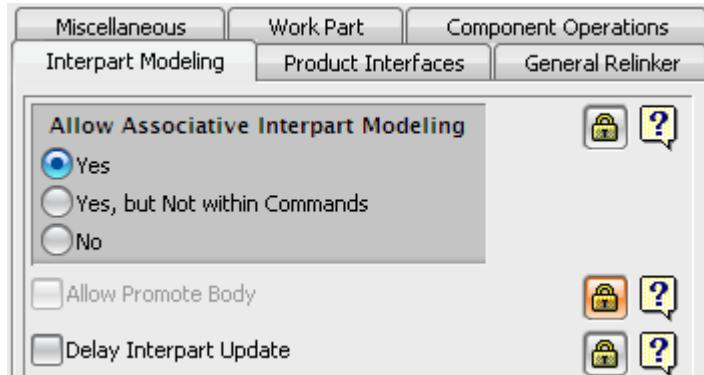


System administrators or managers at a higher level can lock the settings of customer defaults that they do not want anyone at the lower levels to change. The Group level can lock customer defaults at the User level, and the Site level can lock defaults at the Group and User levels. You cannot lock defaults at the User level.

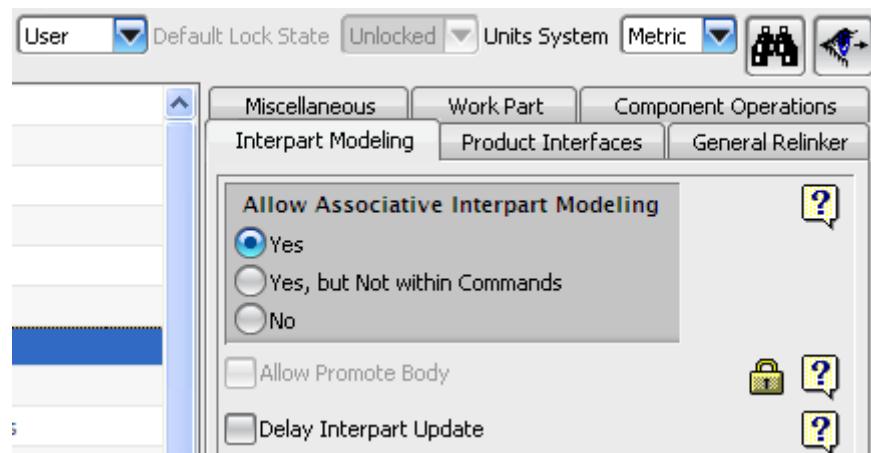
When a lock is active not only is the text de-emphasized but value change is prohibited. Even if the site (or a lower) DPV file is writable the value of a locked default can not be changed until the lock icon has been toggled off for the given default).



For example, to lock out the ability to create promoted bodies, the manager clicks the lock beside promotions at the site or group level. The icon changes color and the text is de-emphasized.



At the user level, that default is de-emphasized and a padlock is displayed beside it.



The manager can use the Default Lock State option to set the global locked status for all of the customer defaults on all defaults pages. This allows strategies like *All are locked except...* or *All are unlocked except...* instead of requiring the assertion of 5000+ individual locks.

Locks at the group level change color and the text is de-emphasized.

The user then sees all options for Site Standards de-emphasized and padlocked. This prevents Site Standards from being changed at the user level.

Setting Customer Defaults

Customer defaults have as-shipped default settings that are hard-coded. When you change defaults at any level (assuming you have write permission and the levels are defined) a file is created to save the settings. By default the file is called nx75_user.dpv, nx75_group.dpv, or nx75_site.dpv.

Only the defaults that are *changed from the hard-coded settings* are saved, thus the DPV files can be *very small* in size.

Customer defaults files are defined by environment settings. These are typically set in ugii_env.dat on Windows systems; however, the administrator may prevent a user from spoofing these settings by creating a file named ugii_env.master in the UGII directory where NX is installed to define these particular environment settings. When this file exists any attempt to redefine the environment variables will be ignored.



When you change defaults the changes are NOT effective immediately. They will be in effect the next time NX is started.

Customer Defaults environment variables

To set up a User, Group, or Site level, you must define the appropriate environment variable with a directory. You must first create a directory named *startup* where you want to store the customer defaults file for that level.

Level	Variable	Defaults File (in the startup directory)
User	UGII_USER_DIR	nx75_user.dpv
Group	UGII_GROUP_DIR	nx75_group.dpv
Site	UGII_SITE_DIR	nx75_site.dpv

If you are already using the UGII_USER_DIR environment variable for other purposes, you can use the UGII_LOCAL_USER_DEFAULTS environment variable. When you define the environment variable, you must point it to the .dpv file you will use (instead of just the directory, as done with the other environment variables).



If both of these environment variables are defined, the system uses the UGII_LOCAL_USER_DEFAULTS environment variable to define the customer defaults User level. NX is shipped with this variable defined, so if you want to use a common user directory (i.e., the one defined by UGII_USER_DIR), you must remove the definition for UGII_LOCAL_USER_DEFAULTS from your environment variables file.

USER, GROUP, and SITE directories

There is a standard structure for customer site installation of menu files and shared libraries. This directory structure defines three subdirectories. For the purpose of this discussion only the *startup* folder need exist; however, you might encounter the others if you have site customization.

- startup Contains site-specific menu files, defaults files, and shared libraries of menu actions to be loaded automatically at NX startup to customize Gateway.
- application Contains site-specific files defining menus and shared libraries of menu actions for customizing NX or third-party applications, such as NX Open programs. Loading of each shared library is deferred until you enter the application that names the library on the LIBRARIES statement in the menu file definition for the Application Button for the application. User Tool Definition files, GRIP programs, User Function programs that are referenced by menu file actions.
- udo Contains the shared libraries defining methods for site-specific User Defined Objects (another NX Open topic.)

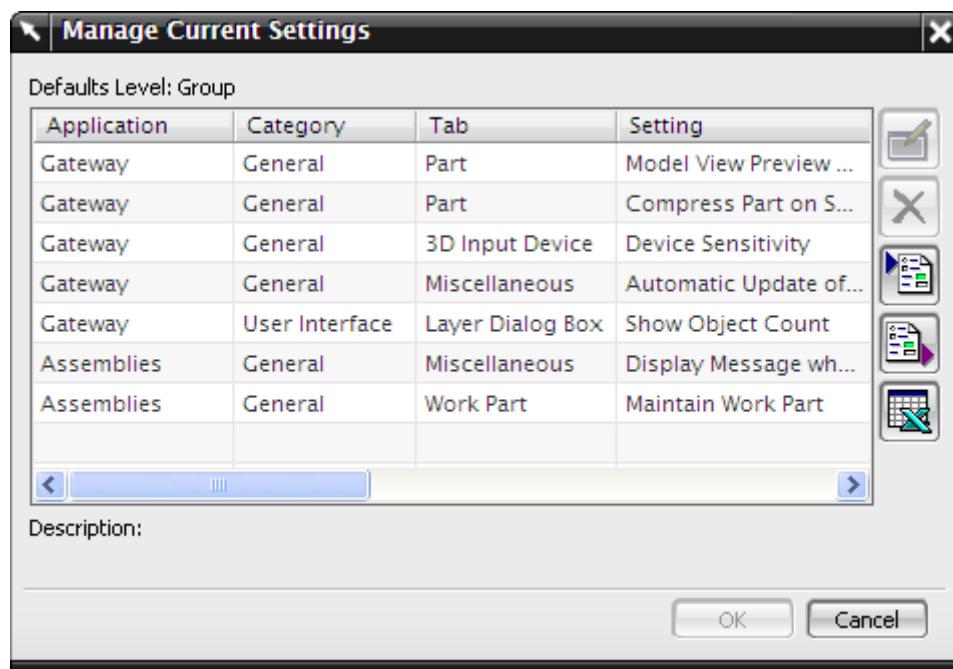
Managing your changes

The DPV files contain only the defaults that are *changed from the hard-coded settings*.

You may review your changes at any time:

- Set the **Defaults Level** to the level you want to examine, Site, Group, or User.
- On the **Customer Defaults** dialog box, click **Manage Current Settings**. 

Here is an example of standard classroom defaults at the group level:



Updating to a new release of NX

To update to a new release, you need only define the DPV files you want to use at whatever levels your organization uses.

When you receive the new software use **Import Defaults** to validate your previous settings against the new release. 

Importing Customer Defaults values file: <full path specification of DPV file.>

Total settings and locks imported: 10

Total settings rejected due to values not valid in this release: 0

Total settings rejected due to values being locked at the higher level: 0

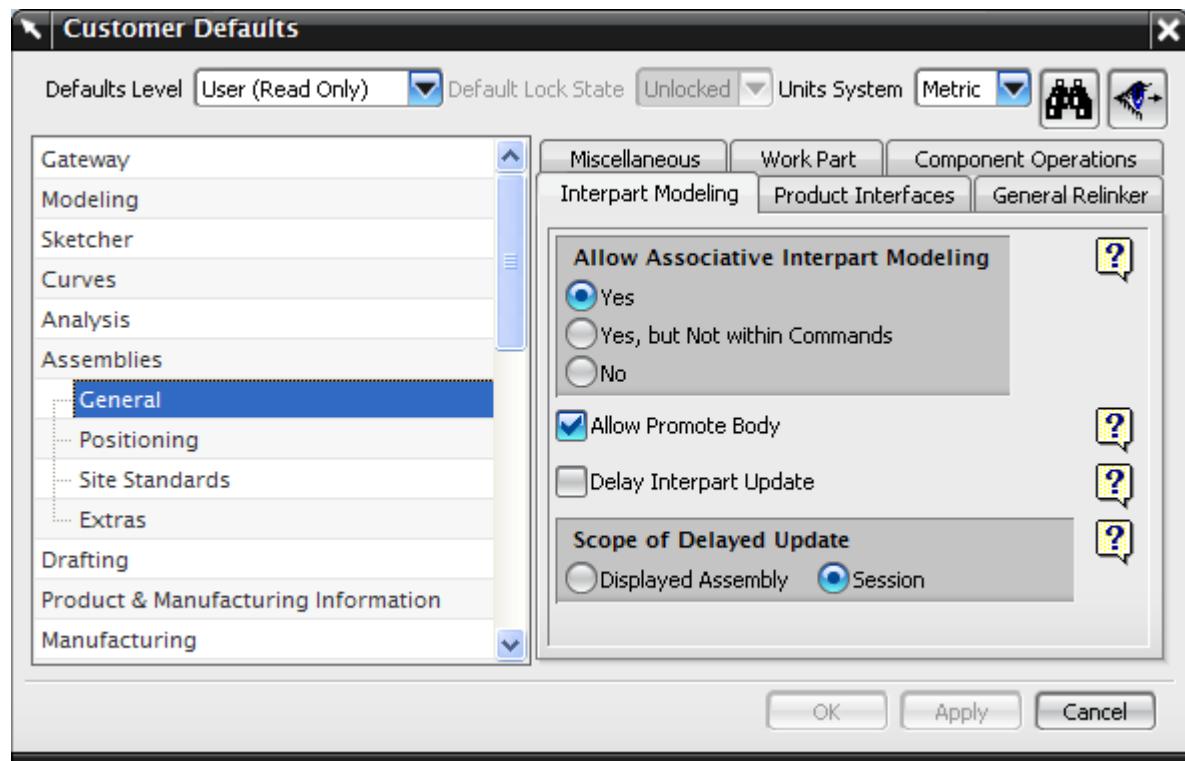
Total settings already set to the same value and lock status: 0

Total settings not recognized in this release: 0

Interpart Modeling

Interpart Expressions and the Wave Geometry Linker can be disabled by changing the setting **Allow Associative Interpart Modeling** on the Assemblies, General, Interpart Modeling page.

Promotion of Bodies feature can be enabled by changing the setting **Allow Promote Body** on the Assemblies, General, Interpart Modeling page.



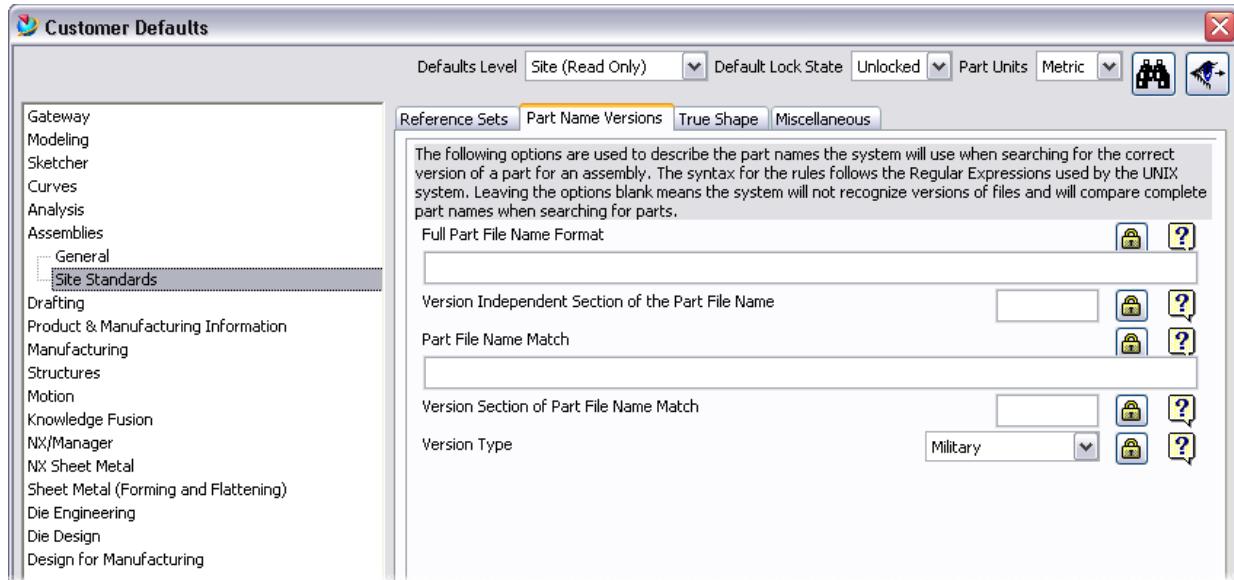
Settings for **Delay Interpart Update** are also on this page.

File Versioning

We recommend using Teamcenter Integration for NX for data management. If it is not available, the Assemblies Site Standards defaults offer an alternative approach.

Versioning rules will enable the system to load the latest version of components in any assembly based on a file naming scheme established at your company. When the rules are defined in the customer defaults file, the "Load Latest" option must also be turned on in the load options.

File versioning is controlled from the Assemblies, Site Standards, Part Name Versions page of the customer defaults dialog.



To define the versioning rules, you must define the portion of the filename that is the core (never changes) and which portion is the revision.

Regular Expressions

Version rules are defined using a modified form of "Regular Expressions". It is beyond the scope of this class to teach regular expression formatting.

There are several "Special Characters" that may be used in the format of your version rules. They are listed below.

? = 0 or 1

*	=	0 or more
+	=	1 or more
.	=	Any character
	=	Or (as in this OR that)
()	=	Define a section
[]	=	Define a set
-	=	Range

The use of Sets is crucial to your ability to establish correct version rules. A few examples are shown below.

[a-z0-9]	=	One lower case letter or number
[a-z0-9] +	=	One or more lower case letters or numbers in any order
[a-z]+[0-9] +	=	One or more lower case letters followed by one or more numbers
[a-hj-np-z]	=	Any lower case letter except i and o
[mejx]	=	Any one of these letters

File Versioning example

In the following example, the part names consist of a core section of lower case letters followed by an underscore and a revision section defined by a revision number. Below are three versions of the same part.

abc_1.prt

abc_2.prt

abc_3.prt

To define the rules that describe the above naming convention, the customer defaults must be modified from the Assemblies, Site Standards, Part Name Versions page of the dialog. Here is an example of how it would need to be set up for a specific versioning scheme.

Full Part File Name Format:

You must specify the format of the part name in terms of regular expression pattern matching. Each set of Parentheses represents a section.

Full Part File Name Format: `([a-z]+_)([0-9])`

The first section is `([a-z]+_)`. The `[a-z]` is any lower case letter and the `"+"` means that one or more letters are allowed. The `"_"` means that the letters will always be followed by an underscore character.

The next section `([0-9])` is any number and that there can be only one digit. (If you wanted to allow more digits, you would follow it with a `"+"`.)

Version Independent Section of the Part File Name:

This determines what portion of the file is the *core* portion of the file name (does not change). Based on the pairs of parentheses, you enter a backslash and the section number.

Version Independent Section of the Part File Name: `/1`

The first section is the core portion of the filename, it will never change.

Part File Name Match:

Specify which section of the file name must match and which section may vary. This is a bit repetitive, but necessary.

Part File Name Match: `/1([0-9])`

The portion of the filename that must match is in the first section. The portion that is allowed to vary may be any number.

Version Section of Part File Name Match:

Which portion of the version section is actually the version. This is also determined by sets of parenthesis in the revision section.

Version Section of Part File Name Match: /1

In the version section, the first set of parenthesis indicates version.

Version Type:

The versioning scheme being used. Available choices are; Military, Numeric, Reverse Numeric, Alphabetic, Reverse Alphabetic, Alphanumeric.

Version Type: Numeric

A number sequence starting with 1 and progressing to larger numbers.



In an Alphanumeric sort, versioning will sort letters before numbers.
This is a different than most normal sort algorithms.

Quantifiers

Quantifiers can be specified to allow only a certain number of characters. Instead of using [0-9][0-9][0-9] to represent three digits, it can be written with a quantifier as [0-9]{3} The list of quantifiers is shown below.

{n,m} At least **n** and no more than

m

{n,} At least **n**

{,m} May have **0**, but no more
than **m**

{m} Exactly **m**

Appendix

D Additional assembly topics

D

This appendix describes additional commands to create, edit, and use assemblies.

Remember Assembly Constraints overview

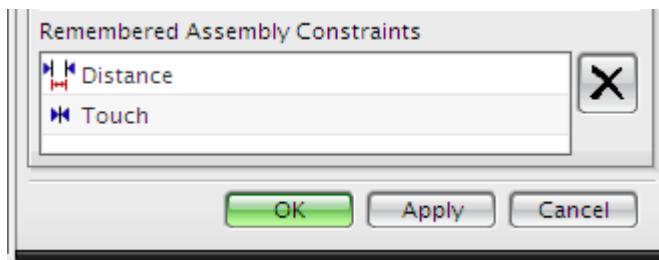
Use the **Remember Assembly Constraints** command to save selected Assembly Constraints that affect the position of a component. When you add that component to a different assembly, the remembered constraints are available to help you position the component.



Recording the constraints will modify the component part. To use these remembered constraints in future sessions, the component part must be saved.



Remembered constraints can be deleted from a component in the **Component Properties** dialog box on the **Part File** page.



Where do I find it?

Application	Assemblies
Toolbar	Assemblies® Remember Assembly Constraints 
Menu	Assemblies® Component Position® Remember Assembly Constraints

Remember assembly constraints

1. On the **Assemblies** toolbar, click **Remember Assembly Constraints** .

The **Remembered Constraints** dialog box opens.

D

2. While **Select Component**  is active, select the component whose positioning constraints you want to remember.



3. While **Select Constraints**  is active, select one or more positioning constraints that you want to save with the component.

4. Click **OK** or **Apply**.

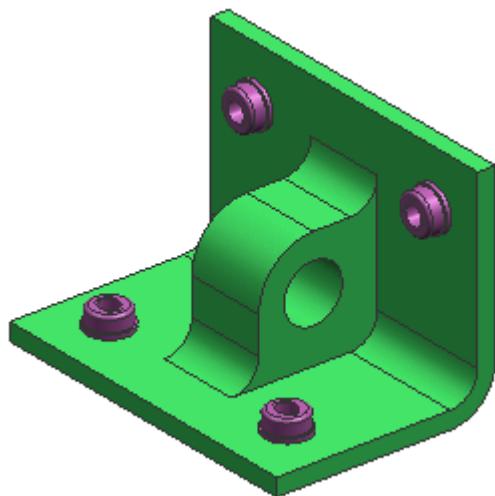
The selected constraints are saved when you save the component. They are available to help you position the component when you add it to any assembly.

When you add a component with remembered constraints to an assembly, the **Redefine Constraints** dialog box appears, which lets you complete the constraints by selecting objects in other components.

Activities: Remember constraints

In the *Appendix: Adding and constraining components* section, do the activity:

- *Remember assembly constraints*



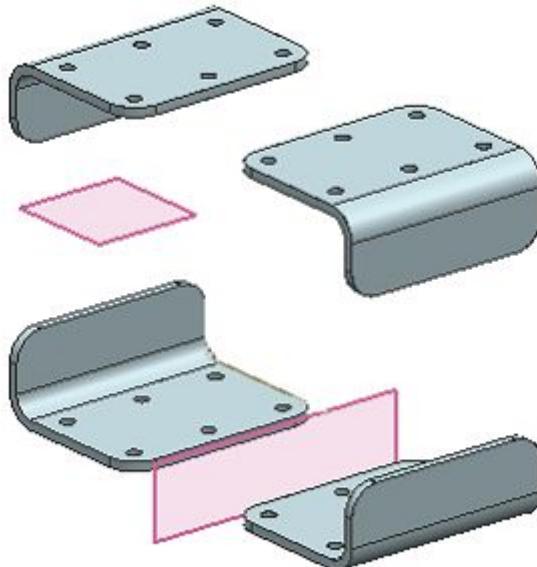


Mirror Assemblies Wizard overview

Use the **Mirror Assemblies** command to:

- Create associative or nonassociative mirrored components in an assembly.
- Position new instances of the same parts at mirror locations.
- Create new parts that contain linked mirror geometry.

D



Many assemblies represent one side of a fairly symmetric larger assembly. You can create one side of your assembly and create a mirrored version to form the other side of your assembly.

You can mirror an entire assembly, or you can select individual components to be mirrored. You can also specify components to be excluded from the mirrored assembly. Each mirrored body is added to the reference set of its source body.

You can specify the plane of symmetry plane of symmetry that is used to mirror the component.

You can create a mirrored component in one of the following ways:

- Create mirror geometry. When you do this, the software creates a new part file that contains the linked mirror body features for all the solid geometry in the original component. It also adds the new mirror part to the assembly as a component.
- Perform a mirror reuse and reposition. This is a two-dimensional reposition of a new instance of the selected component. When you do this, the software creates a new instance of the original part. It does not create any new files.

Before you create the mirrored assembly, you can preview it and make corrections using the **Mirror Review** step.

Where do I find it?

Application	Assemblies
Prerequisite	An assembly must be the work part.
Toolbar	Assemblies® Mirror Assembly 
Menu	Assemblies® Components® Mirror Assembly



Create a Mirror Assembly

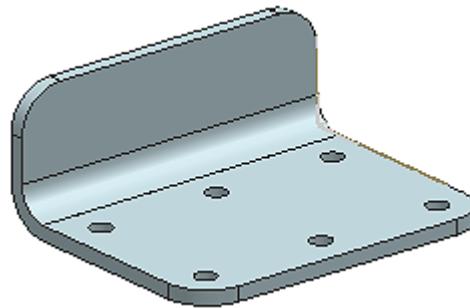
1. On the **Assemblies** toolbar, from the **Components Drop-down** list, select **Mirror Assembly** , or choose **Assemblies® Components® Mirror Assembly**.

The **Mirror Assemblies Wizard** welcome page opens.

2. Click **Next**.

The **Select Components** page opens.

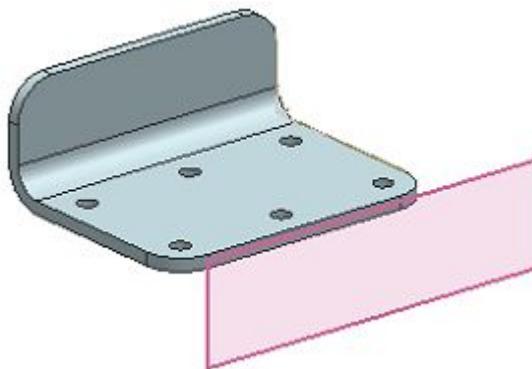
3. Select the components to mirror and click **Next**.



The components must be children of the work assembly.

The **Select Plane** page opens.

4. Select an existing plane or click **Create Datum Plane** , and create a datum plane.



5. Click **Next**.

The **Mirror Setup** page opens. Selected components are listed in the right-hand panel.

6. (Optional) Select a different mirror type for each component. Select a component from the list of components in the right panel and click any of the following:

- Click **Associative Mirror**  to create an associative opposite-side version of a component and create new parts.
- Click **Non Associative Mirror**  to create an nonassociative opposite-side version of a component and create new parts.
- Click **Exclude**  to exclude the selected component.

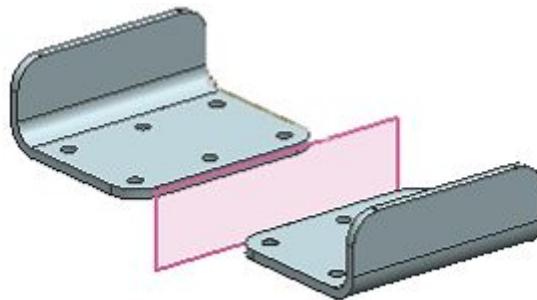
Reuse and Reposition is the default mirror type.

7. Click **Next**.

The mirrored components appear in the graphics window. The **Mirror Review** page opens.

8. (Optional) Make corrections before finishing the operation. You can do any of the following:

- Change a component's mirror type from **Reuse and Reposition** to **Associative Mirror** or **Non Associative Mirror**.



- Select components and click **Exclude**
- Click **Cycle Reposition Solutions**

9. Click **Next**.

The **Naming Policy** page opens. If no component uses the **Mirror Geometry** type, you can click **Finish**. Otherwise, proceed to the next step.

10. Specify the naming policy that you want to use for these opposite-side parts, and specify a directory for the parts.

You can add a prefix or suffix to the name used by their source parts.

11. After you specify the naming policy and the directory for the parts, do the following:

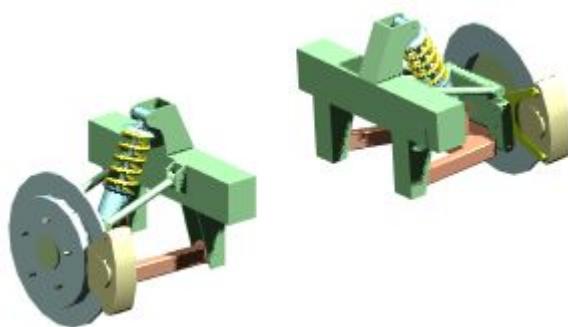
- Click **Next** to open the **Name New Part Files** page and to review the names that the software applied to your new opposite-side parts.
- Double-click a name to change it.
- In the **Rename New Part File** dialog box, specify a new unique name.
- Click **OK**.

12. Click **Finish**.

Activities: Mirror Assembly

In the *Appendix: Interpart geometry* section, do the activity:

- *Mirror Assembly*

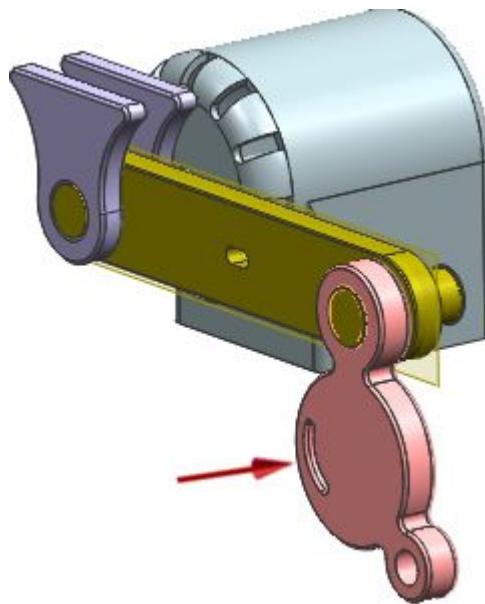




Create a WAVE-linked mirror body with timestamp

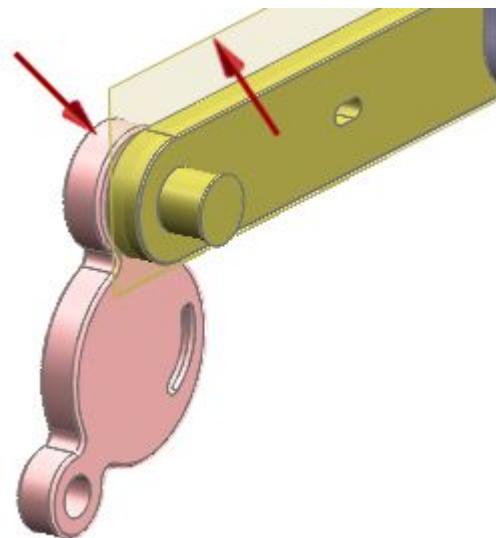
This example shows how to create an associative WAVE linked mirror body with a fixed timestamp constraint.

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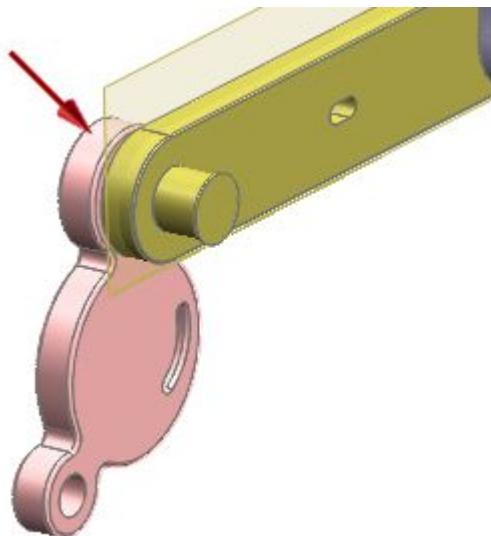
1. In the **Assembly Navigator**, right-click the component you want to have own the linked geometry and choose **Make Work Part**.

In this case, the component is empty. You want the weight component mirrored about the center plane of the bar component.

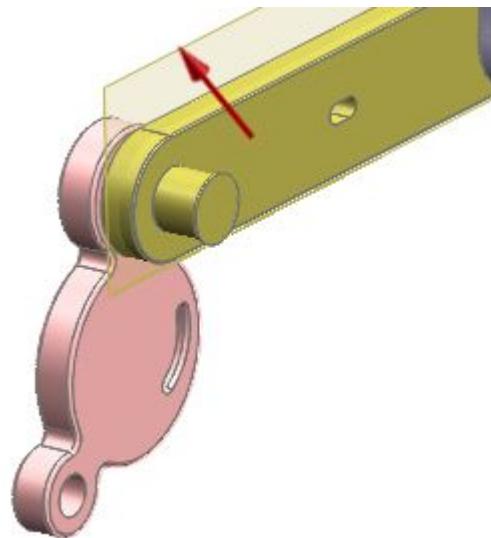


2. On the **Assemblies** toolbar, click **WAVE Geometry Linker** , or choose **Insert→Associative Copy→WAVE Geometry Linker**.

3. In the **WAVE Geometry Linker** dialog box, from the **Type** list, select **Mirror Body** .
4. In the graphics window, from the parent part, select the body to be mirrored.



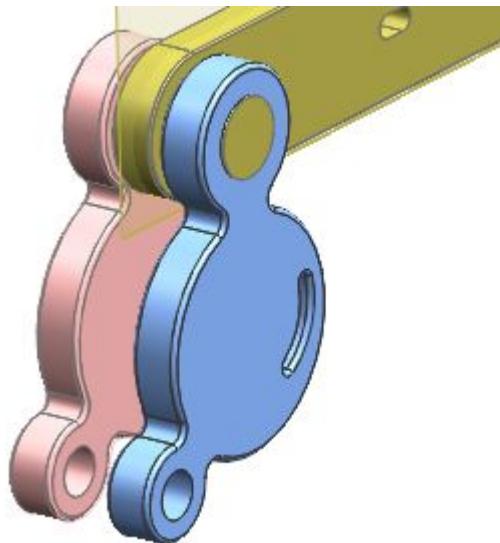
5. Click **Select Mirror Plane** .
6. In the graphics window, from the parent part, select the datum plane.



The new mirrored body is associative to the parent parts of the body and the datum plane.

7. In the **Settings** group, ensure that the **Associative** check box is selected.

8. In the **Settings** group, ensure that the **Fix at Current Timestamp** check box is selected.
 - 💡 Any new features added to the source geometry are not reflected in the mirrored body.
9. (Optional) Select the **Use Display Properties of Parent Part** check box to retain the display properties of the parent part.
10. Click **OK**.

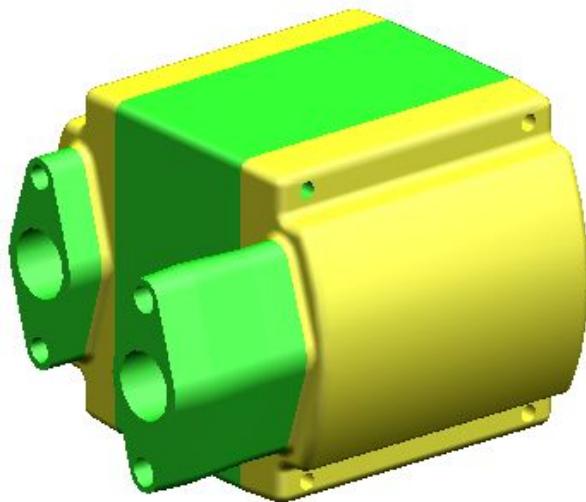


💡 The mirrored body is an exact duplicate of the source geometry visually and in regards to position. You can move the mirrored body; it is position-independent.

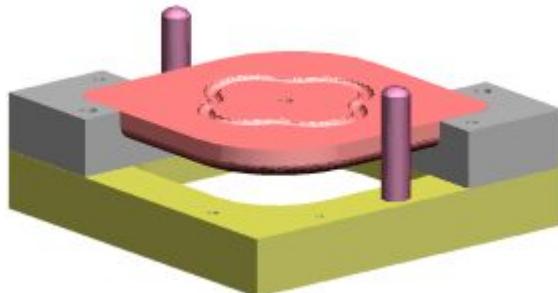
Activities: WAVE Geometry Linker

In the *Appendix: Interpart geometry* section, do the activities:

- *WAVE Geometry Linker - Mirror Body*



- *Design in context of an assembly*





Replace with Independent Sketch

Use the **Replace with Independent Sketch** command to replace a linked curve feature with an independent sketch.

This command will replace a WAVE linked sketch or a two-dimensional linked composite curve with an identical sketch that is completely independent of the parent.

D

Why should I use it?

Use this command when you no longer want a design to be controlled by its master layout.

For example, during the design and development of a product, you may have a master layout to control the design of many component parts. When the designs are released for manufacturing, you do not want the components to change if the master layout is changed. To do this, you can break the link and replace the linked sketch with an independent sketch. An additional advantage is that the replaced independent sketch has its own constraints that you can modify.

Where do I find it?

Application	Modeling
Toolbar	Edit Feature® Replace with Independent Sketch 
Menu	Edit® Feature® Replace with Independent Sketch
Part Navigator	Right-click a Linked feature® Replace with Independent Sketch

Hole Series

Use the **Hole Series** type to create a set of related holes. You can use this type to mount a fastener across multiple solids.

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You can create holes through:

- Multiple bodies in the work part.
- Multiple bodies in an assembly.

Where do I find it?

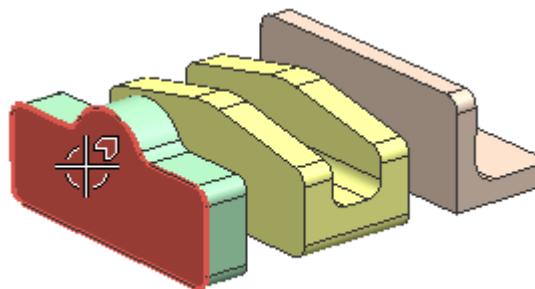
Application	Modeling
Toolbar	Feature® Hole 
Menu	Insert® Design Feature® Hole
Location in dialog box	Type list® Hole Series



Create a Hole Series feature

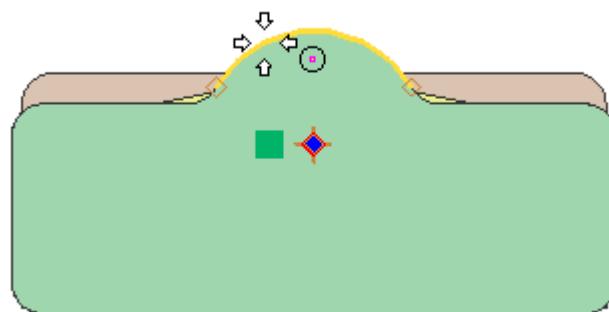
This example shows how to create a **Hole Series** feature through three parts in an assembly.

1. On the **Feature** toolbar, click **Hole** or choose **Insert® Design Feature® Hole**.
2. In the **Hole** dialog box, from the **Type** list, select **Hole Series**.
3. In the graphics window, click on the near face of the component for the start face.



Specified face as the sketch plane

4. Highlight the edge of the arc and select the arc center.



Specify Arc Center

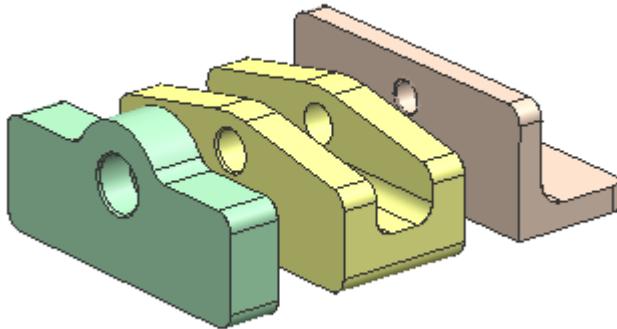
5. In the **Sketch Point** dialog box, click **Close**.
6. Click **Finish Sketch** .
7. In the **Specification** group, specify values for the **Start**, **Middle**, and **End** hole dimensions.

8. Click **OK**, to create a **Hole Series** feature in the assembly.



The **Hole Series** feature is listed in the **Assembly Navigator** as a feature and is also listed in each component part as a linked feature.

D





Editing a Hole Series feature

You can edit a Hole Series feature from the assembly or one of the components.

- When you edit from the assembly, all links are .
- When you edit from the component you can choose between these options:
 - **Break Link from Hole Series:** Breaks the associative link from the parent hole series. The linked hole is converted to a standalone Hole feature. The rest of the series is unaffected. You cannot recreate the link to the parent.
 - **Edit Hole Series:** Edits the parent Hole Series feature from a linked child Hole feature.
 - The **Edit Hole Series** option is only available when the assembly that contains the Hole Series feature is open.
 - You cannot edit the Hole Series feature if one of the parts is not open, or if you cannot make the part that contains the Hole Series feature the work part.

D

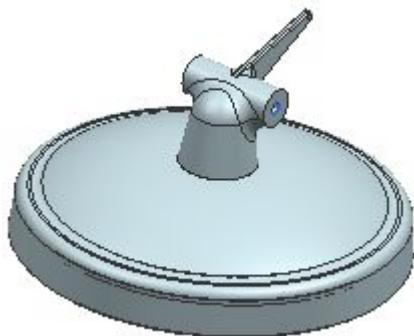
In the **Settings** group, the **Extend Start** tolerance is used to adjust the tolerance between the start face and the hole feature.

If you move the placement face of the hole you need to adjust the tolerance so the hole can extend the proper distance.

Activities: Create and edit a hole series

In the *Appendix: Interpart geometry* section, do the activity:

- *Create and edit a hole series*





Promote Body overview

Use the **Promote Body** command to promote a body from a loaded assembly component to the level of the assembly.

D

You do not need write access to the part containing the original, or the base body, in order to promote it. This can be especially important when working in a concurrent engineering environment.

After you promote a body:

- You can perform operations on it, such as adding features, performing Boolean operations between it and other bodies, and so on. The effects of these operations are only visible at the level of the assembly work part in which the promotion is created, and in any other assembly that references that part.
- You cannot access the features that make up the base body at the assembly level. For example, you cannot suppress a feature of the base body, or change its parameters and have those changes occur only at assembly level. You must make such changes to the base body. They will be visible in the promoted body, since it is associative to the base body.
- Any subsequent changes to the base body are reflected in the promoted body as the promoted body is associative to the base body. However, changes made to a promoted body do not affect the base body.



You must be careful when working "in context" (that is, with the work part different than the displayed part) with promoted bodies. If the assembly is the displayed part and the component is the work part, the promoted body is still displayed and this blocks the display of the base body. You should change the displayed part to the component to work on the base body.



You should not promote a body that depends on another body in the same part. A dependent body is created by features that generate a new body, such as a mirror body feature, an extract face feature or a midsurface feature. If you promote a dependent body and then suppress or delete the parent body or feature, you may get internal errors.

Where do I find it?

Application	Modeling
Prerequisite	The work part must be an assembly.
Toolbar	Feature® Associative Copy Drop-down® Promote 

Menu	Insert® Associative Copy® Promote Body
------	---

Customer defaults for **Promote Body** are at:

Menu	File® Utilities® Customer Defaults
Location in dialog box	Customer Defaults® Assemblies® Interpart Modeling® Allow Promote Body

Activities: Promotions

In the *Appendix: Interpart geometry* section, do the activity:

- *Promotions*



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Assembly Cut overview

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Use the **Assembly Cut** command to associatively subtract one or more tools from one or more bodies at any assembly level part.

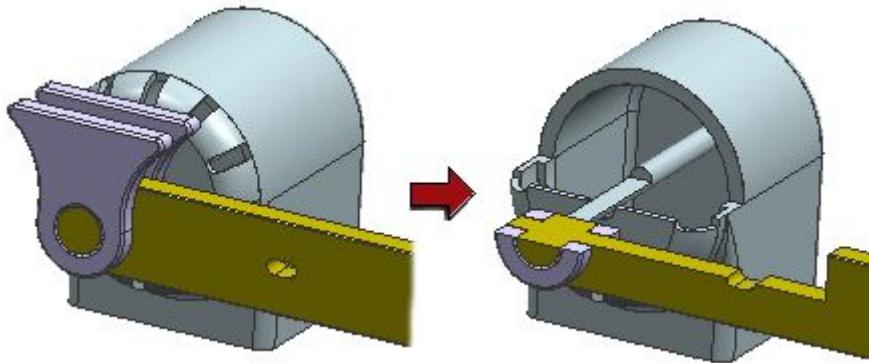
You can create:

- Holes and other cutout shapes that you do not model in the piece part because they are made in varying locations as the component is assembled in different assemblies and locations.
- Section cuts to illustrate your assembly without affecting the master model.

All target bodies affected by the cut are copied and displayed in the assembly. To use **Assembly Cut** the assembly part must be the work part.



The image on the left shows the assembly pre-cut and the image on the right displays the copied target components post-cut.



Where do I find it?

Application	Modeling
Toolbar	Feature Operation→Assembly Cut
Menu	Insert→Combine Bodies→Assembly Cut

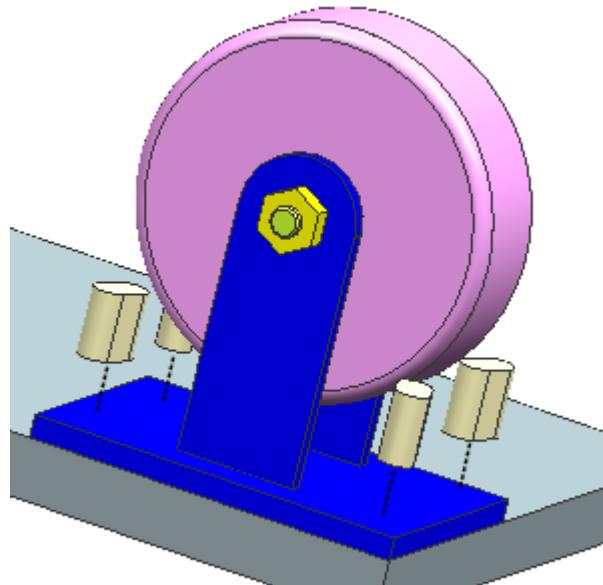


Create an Assembly Cut

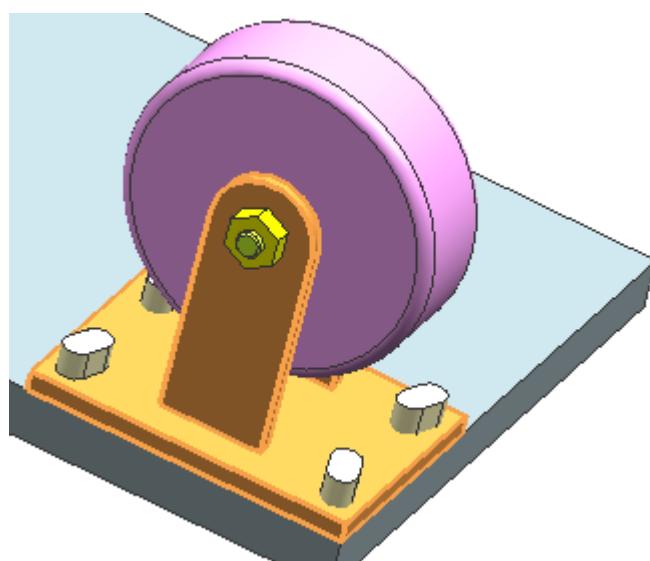
This example shows how to use the **Assembly Cut** command to create slots at potential plug weld locations.

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1. Make sure your assembly is the work part.



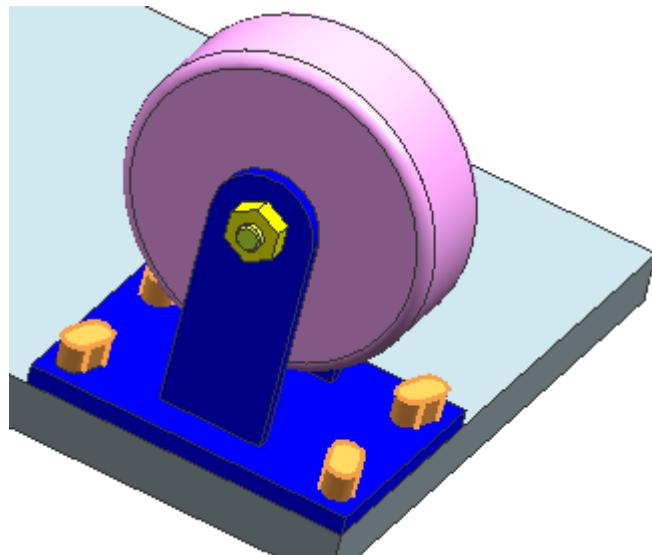
2. On the **Feature Operations** toolbar, click **Assembly Cut** , or choose **Insert→Combine Bodies→Assembly Cut**.
3. Select one or more target bodies.



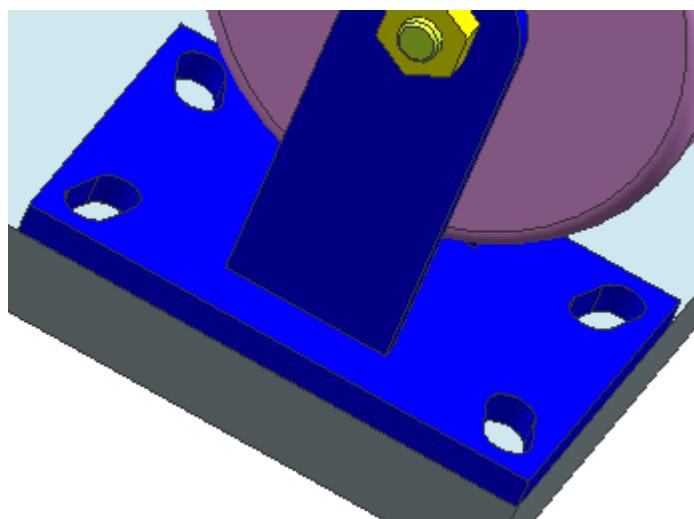
4. In the **Assembly Cut** dialog box, In the **Tool** group, click **Select Body**, in the graphics window select one or more tool solid bodies.



Tool bodies must be solid bodies. The operation fails if you select a sheet body as a tool.



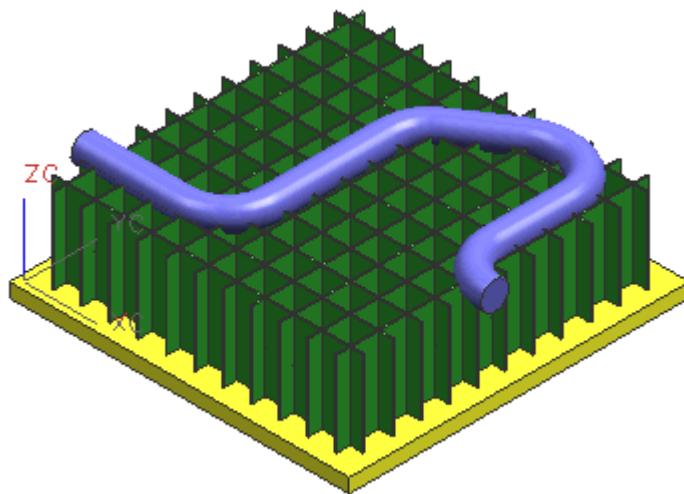
5. If you want the tool solid to remain visible after the operation is complete, clear the **Hide Tool** check box.
6. Click **OK**.



Activities: Assembly Cut

In the *Appendix: Interpart geometry* section, do the activity:

- *Assembly Cut*



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WAVE

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STUDENT PROFILE

In order to stay in tune with our customers we ask for some background information. This information will be kept confidential and will not be shared with anyone outside of Education Services.

Please Print...

Your Name

U.S. citizen Yes No

Course Title/Dates

/ thru

Hotel/motel(s) while training

Planned departure time after class

Employer

Location

Supervisor/manager

(Emergency) Phone

Your job title/responsibilities

/

Industry: Auto Aero Consumer products Machining Tooling Medical Other

Types of products/parts/data that you work with

Platform (operating system)

Reason for training

Please verify/add to this list of training for NX, I-deas, Imageware, Teamcenter Mfg., Teamcenter Engineering, Teamcenter Enterprise, Tecnomatix or Dimensional Mgmt./Visualization. Medium means Instructor-lead (IL), On-line (OL), or Self-paced (SP)

Software	From Whom	When	Course Name	Medium

Other CAD/CAM/CAE /PDM software you have used _____

Please "check"! your ability/knowledge in the following...

Subject	None	Novice	Intermediate	Advanced
CAD modeling	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
CAD assemblies	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
CAD drafting	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
CAM	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
CAE	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
PDM – usage	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
PDM – system management	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
PDM – customization	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>

Thank you for your participation. We hope your training experience will be an outstanding one.

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Course Agenda

Introduction to NX for Experienced Users

Introduction to NX for Experienced Users with Teamcenter Integration

Monday **Morning**

- Lesson 1. NX part files
- Lesson 2. The NX user interface

Afternoon

- Lesson 3. Sketching
 - Lesson 4. Constraining and using sketches
-

Tuesday **Morning**

- Lesson 5. Datum features
- Lesson 6. Swept features
- Lesson 7. Swept feature options

Afternoon

- Lesson 8. Trim Body
 - Lesson 9. Hole features
 - Lesson 10. Expressions
-

Wednesday **Morning**

- Lesson 11. Coordinate systems
- Lesson 12. Part Navigator
- Lesson 13. Associative copies

Afternoon

- Lesson 14. Face operations
 - Lesson 15. Edge operations
 - Lesson 16. Basic freeform
-

Thursday **Morning**

- Lesson 17. Introduction to Assemblies
- Lesson 18. Adding and constraining components
- Lesson 19. Reference Sets

Afternoon

- Lesson 20. Top-down assemblies
 - Lesson 21. Assembly arrangements
 - Lesson 22. Interpart geometry
-

Friday **Morning**

- Lesson 23. Interpart references
- Lesson 24. Component arrays
- Lesson 25. Reuse Library

Afternoon

- Lesson 26. Revise and replace components
- Lesson 27. Introduction to Drafting
- Lesson 28. Editing models

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Accelerators

You can list the following keyboard accelerators in NX.
Choose Information→Custom Menubar→Accelerators.

Command	Accelerator
File→New...	Ctrl+N
File→Open...	Ctrl+O
File→Save	Ctrl+S
File→Save As...	Ctrl+Shift+A
File→Plot...	Ctrl+P
File→Execute→Grip...	Ctrl+G
File→Execute→Debug Grip...	Ctrl+Shift+G
File→Execute→NX Open...	Ctrl+U
File→Finish Sketch	Ctrl+Q or Q
Edit→Undo	Ctrl+Z
Edit→Redo	Ctrl+Y
Edit→Cut	Ctrl+X
Edit→Copy	Ctrl+C
Edit→Paste	Ctrl+V
Edit→Delete...	Ctrl+D or Delete
Edit→Selection - Select All	Ctrl+A
Edit→Object Display...	Ctrl+J
Edit→Show and Hide→Show and Hide... (by type)	Ctrl+W
Edit→Show and Hide→Immediate Hide...	Ctrl+Shift+I
Edit→Show and Hide→Hide...	Ctrl+B
Edit→Show and Hide→Show...	Ctrl+Shift+K
Edit→Show and Hide→Show All	Ctrl+Shift+U
Edit→Show and Hide→Invert Shown and Hidden	Ctrl+Shift+B
Edit→Move Object	Ctrl+T
Edit→Sketch Curve→Quick Trim...	T
Edit→Sketch Curve→Quick Extend...	E
View→Operation→Fit	Ctrl+F
View→Operation→Zoom...	Ctrl+Shift+Z
View→Operation→Rotate...	Ctrl+R
View→Section→Edit Work Section...	Ctrl+H
View→Visualization→High Quality Image...	Ctrl+Shift+H
View→Layout→New...	Ctrl+Shift+N
View→Layout→Open...	Ctrl+Shift+O
View→Layout→Fit All Views (only with multiple views)	Ctrl+Shift+F
View→Information Window	F4
Hide or show the current dialog box	F3
View→HD3D Tools UI	Ctrl+3
View→Move Clip Left	Shift+F1
View→Move Clip Right	Shift+F2

View→Full Screen or View→Normal Screen	Alt+Enter
View→Orient View to Sketch	Shift+F8
View→Reset Orientation	Ctrl+F8
Insert→Sketch Curve → Profile...	Z
Insert→Sketch Curve → Line...	L
Insert→Sketch Curve → Arc...	A
Insert→Sketch Curve → Circle...	O
Insert→Sketch Curve → Fillet...	F
Insert→Sketch Curve → Rectangle...	R
Insert→Sketch Curve → Polygon...	P
Insert→Sketch Curve → Studio Spline...	S
Insert→Sketch Constraint → Dimension → Inferred...	D
Insert→Sketch Constraint → Constraints...	C
Insert→Design Feature→Extrude...	X
Insert→Surface→Four Point Surface...	Ctrl+4
Insert→Mesh Surface→Studio Surface...	N
Insert→Sweep→Variational Sweep...	V
Format→Layer Settings...	Ctrl+L
Format→Layer Visible in View...	Ctrl+Shift+V
Format→WCS→Display	W
Tools→Expression...	Ctrl+E
Tools→Update→Make First Feature Current	Ctrl+Shift+Home
Tools→Update→Make Previous Feature Current	Ctrl+Shift+Left Arrow
Tools→Update→Make Next Feature Current	Ctrl+Shift+Right Arrow
Tools→Update→Make Last Feature Current	Ctrl+Shift+End
Tools→Journal→Play...	Alt+F8
Tools→Journal→Edit	Alt+F11
Tools→Macro→Start Record...	Ctrl+Shift+R
Tools→Macro→Playback...	Ctrl+Shift+P
Tools→Macro→Step...	Ctrl+Shift+S
Tools→Movie→Record	Alt+F5
Tools→Movie→Pause	Alt+F6
Tools→Movie→Stop	Alt+F7
Tools→Customize...	Ctrl+1
Information→Object...	Ctrl+I
Analysis→Curve→Refresh Curvature Graphs	Ctrl+Shift+C
Preferences→Object...	Ctrl+Shift+J
Preferences→Selection...	Ctrl+Shift+T
Application→Modeling...	M or Ctrl+M
Application→Shape Studio...	Ctrl+Alt+S
Application→Drafting...	Ctrl+Shift+D
Application→Manufacturing...	Ctrl+Alt+M
Application→NX Sheet Metal...	Ctrl+Alt+N
Application→Flexible Printed Circuit Design...	Ctrl+Alt+P
Help→On Context...	F1
Refresh	F5

Zoom	F6
Rotate	F7
Orient View-Trimetric	Home
Orient View-Isometric	End
Orient View-Top	Ctrl+Alt+T
Orient View-Front	Ctrl+Alt+F
Orient View-Right	Ctrl+Alt+R
Orient View-Left	Ctrl+Alt+L
Snap View	F8

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PLM Software
Evaluation – Delivery

Name: _____

Course #: _____

Start Date: _____

Through: _____

Please share your opinion in all of the following sections with a "check" in the appropriate box:

Instructor:

Instructor: If there were 2 instructor

s, please evaluate the 2nd instructor with "X's"

1. Clearly explained the course objectives.....
2. Was knowledgeable about the subject.....
3. Answered my questions appropriately.....
4. Encouraged questions in class.....
5. Was well spoken and a good communicator.....
6. Was well prepared to deliver the course.....
7. Made good use of the training time.....
8. Conducted themselves professionally.....
9. Used examples relevant to the course and audience.....
10. Provided enough time to complete the exercises.....
11. Used review and summary to emphasize important information.....
12. Did all they could to help the class meet the course objectives.....

	STRONGLY DISAGREE	DISAGREE	SOMEWHAT DISAGREE	SOMEWHAT AGREE	AGREE	STRONGLY AGREE
1.	<input type="checkbox"/>					
2.	<input type="checkbox"/>					
3.	<input type="checkbox"/>					
4.	<input type="checkbox"/>					
5.	<input type="checkbox"/>					
6.	<input type="checkbox"/>					
7.	<input type="checkbox"/>					
8.	<input type="checkbox"/>					
9.	<input type="checkbox"/>					
10.	<input type="checkbox"/>					
11.	<input type="checkbox"/>					
12.	<input type="checkbox"/>					

Comments on overall impression of instructor(s):

Overall impression of instructor(s)..... Poor Excellent

Suggestions for improvement of course delivery: _____

What you liked best about the course delivery: _____

Class Logistics:

1. The training facilities were comfortable, clean, and provided a good learning environment.....
2. The computer equipment was reliable.....
3. The software performed properly.....
4. The overhead projection unit was clear and working properly.....
5. The registration and confirmation process was efficient.....

<input type="checkbox"/>					
<input type="checkbox"/>					
<input type="checkbox"/>					
<input type="checkbox"/>					
<input type="checkbox"/>					

Hotels: (We try to leverage this information to better accommodate our customers)

1. Name of the hotel _____ Best hotel I've stayed at..
2. Was this hotel recommended during your registration process?..... YES NO
3. Problem? (brief description) _____

SEE BACK

Evaluation - Courseware

Name: _____ Course #: _____
Dates: _____ Through: _____

Please share your opinion for all of the following sections with a "check" in the appropriate box:

	STRONGLY DISAGREE	DISAGREE	SOMEWHAT DISAGREE	SOMEWHAT AGREE	AGREE	STRONGLY AGREE
1. The training material supported the course and lesson objectives.....	<input type="checkbox"/>					
2. The training material contained all topics needed to complete the projects.....	<input type="checkbox"/>					
3. The training material provided clear and descriptive directions.....	<input type="checkbox"/>					
4. The training material was easy to read and understand.....	<input type="checkbox"/>					
5. The course flowed in a logical and meaningful manner.....	<input type="checkbox"/>					
6. How appropriate was the length of the course relative to the material?	<input type="checkbox"/>					
		Too short	Too long	Just right		

Comments on Course and Material: _____

Overall impression of course..... Poor Excellent

Student:

- I met the prerequisites for the class (I had the skills I needed).....
- My objectives were consistent with the course objectives.....
- I will be able to use the skills I have learned on my job.....
- My expectations for this course were met.....
- I am confident that with practice I will become proficient.....

Name (optional): _____ Location/room _____

Please "check" this box if you would like your comments featured in our training publications.
 (Your name is required at the bottom of this form)

Please "check" this box if you would like to receive more information on our other courses and services.
 (Your name is required at the bottom of this form)

Thank you for your business. We hope to continue to provide your training and personal development for the future.