Lesson 25
Assembly Tools

Learning Objective

At the conclusion of this lesson, the user will have a good overall understanding of the tools used in constraining and managing assemblies.

The Assembly tool bar is divided into four sections: Component, Assembly Constraints, Component Management, and Component Viewing.

Assembly components can be individual parts or subassemblies that behave as a single unit. For example, a single-part base plate and a multi-part air cylinder subassembly are both components when placed in an assembly.

To make sure that they are always available when you open the assembly, add the paths for all components to the project file for the assembly.

The behavior and characteristics of a component depend on its origin.

A Mechanical Desktop part placed as a component in an Autodesk Inventor assembly acts much like any assembly component. You can add assembly constraints, set its visibility, and perform other assembly operations. However, you cannot edit the part in Autodesk Inventor.

Each Mechanical Desktop part is linked to the assembly through a special file called a proxy file. The proxy file contains the linking information so that the assembly component updates when you edit the part in Mechanical Desktop.

TIP: If you make extensive changes to any component in an assembly, some assembly constraints may not compute correctly when the Autodesk Inventor assembly file is updated. These constraints must be recreated.

Parts or subassemblies created using another CAD system can be inserted as components in the active assembly. You cannot change the size or shape of external components, but you can customize them by adding features.

Adaptive parts can change size and shape to satisfy assembly design requirements. When an adaptive part is constrained to other assembly components, underconstrained geometry in the adaptive part resizes.

When a part is first placed in an assembly, it is not defined as adaptive in the assembly context. You can create fixed-size geometry, and then place the part in an assembly. Select one occurrence in the assembly and designate it as adaptive.

Most assemblies contain a combination of existing components and components (parts and subassemblies) created in the assembly environment.

When you create components in place, you can use geometry from other parts (such as edges and hole centers) in feature sketches. Parts based on existing geometry are sized and positioned in relation to that geometry. Parts created in place have an automatic mate constraint applied between the part XY sketch plane and the part face you sketch on. You can define a part created in place as adaptive so that its size and shape can adjust as assembly requirements change.

Any part in an assembly may have all of its degrees of freedom removed and be fixed in position, relative to the assembly coordinate system. The origin of a grounded part will not move when you place assembly constraints, but a grounded part can still be designated as adaptive. The features on a grounded, adaptive part can change size or shape although its position is fixed.

TIP: The first component placed in an assembly is automatically grounded, so that subsequent parts may be placed and constrained in relation to it. If necessary, you can remove the grounded status of a part.

Placing the First Component

The first component placed in an assembly should be a fundamental part or subassembly, such as a frame or base plate, on which the rest of the assembly is built.

The first component in an assembly file sets the orientation of all subsequent parts and subassemblies. The part origin is coincident with the origin of the assembly coordinates and the part is grounded (all degrees of freedom are removed).

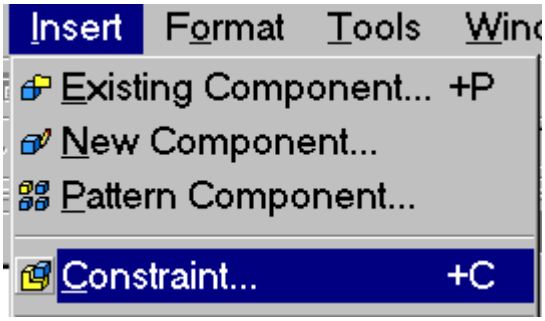
If necessary, you can restore degrees of freedom to the grounded part (the base component) and reposition it. Any components you have constrained to it will also move.

TIP: It is a good idea to place parts and subassemblies in the order in which they would be assembled in manufacturing.

Although there is no distinction in an assembly between components, you can think of the first component you place as the base component because it is usually a fundamental component to which others are constrained. If you place a first component and then want to change to a different base component, you can place a new component, specify it as grounded, and then reconstrain any components you placed earlier, including the first component. Right-click on the first component, clear the Grounded check box, and then constrain it to the new base component.

There is no limit to how many components can be grounded, but most assemblies have only one grounded component. Grounded components are appropriate for fixed objects in assemblies because their position is absolute (relative to the assembly coordinate origin) and all degrees of freedom are removed. Grounded components have no dependencies on other components.

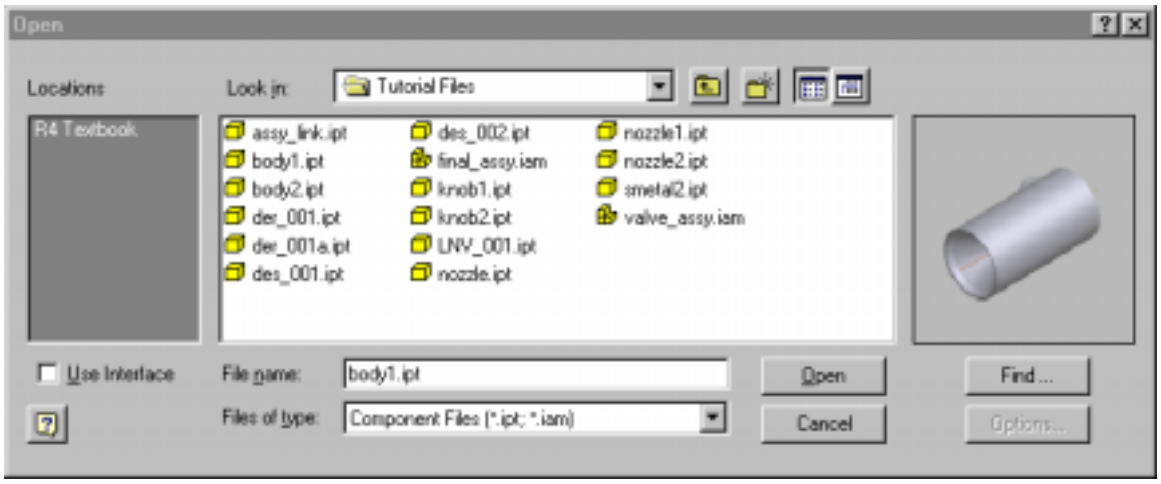
You can use the Move button on the Assembly toolbar to relocate the grounded component. You can drag the component to its new position. The component is grounded in the new location.



Assemblies can also be managed from the Menu. You can place an Existing Component, Create a New Component, Pattern a Component, or Add a Constraint under the Insert Menu.

Place Component

The first component in an assembly is automatically positioned with its origin coincident with the assembly coordinate origin. Additional components are positioned with the cursor, attached at the component center of gravity.



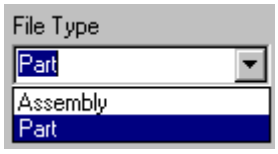
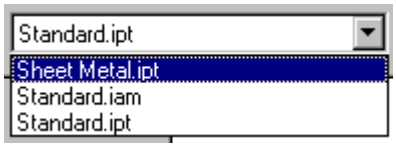
1. Click the Place Component button to choose a component to place.
2. Go to the folder that contains the component, select the component, and click Open.
3. The selected component is placed in the graphics window, attached to the cursor. Select a location and click to place an occurrence of the component.
4. Move the cursor to a different location and click to place a second occurrence, continuing until all occurrences are placed.
5. To quit, right-click and select ‘Done’.

TIP: You can drag and drop a component from Windows Explorer or a browser window, but the component includes undisplayed default work planes that may offset the part from the cursor. Drag and drop places a single instance, unlike multiple occurrences as described above. Whether you place components through the dialog box or drag and drop, use assembly constraints to position components and remove degrees of freedom.

Typing a ‘P’ will initiate the Place Component command.

Create Component

You can create a new part in the context of an assembly file. Creating an in-place part has the same result as opening a part file, with the additional option of sketching on the face of an assembly component or an assembly work plane. To allow the size of the new part to change with assembly requirements, you can designate the part as adaptive and constrain it to fixed geometry in the assembly.

New File Name	Name of the file for the new component
	Part – defines part as a single component Assembly – defines part as a subassembly
	Constrain sketch plane to selected face Enabling will automatically add a mate constraint for the new part/subassembly Disabling means that the part will be “free-floating” and will need to be constrained later
New File Location	Set the subdirectory where the file is to be saved.
	Sets the file extension to be defined to the new file.

Extruded features can start and end on faces of other parts. By default, extruded features with From To or To extents are adaptive.

1. Click the Create Component button.
2. In the dialog box, enter a name for the part and click OK.
3. Select a component face or work plane on which to sketch.
4. If you want to reorient the view to the sketch, click the Look At button.
5. Use the tools on the Sketch toolbar to create a sketch on a selected plane.
6. Select Extrude, Revolve, Loft, or Sweep to create a feature using the new sketch.
7. Continue to select faces on which to sketch and add new features as needed.

When the part is complete, double-click the top-level assembly in the browser to reactivate the assembly environment.

TIP: A mate constraint is automatically placed between the new sketch and the face or work plane. To omit this constraint, clear the check box in the Create Part In-Place dialog box when you create the part file. You can set options on the Adaptive tab of the Options dialog box to control feature termination.

Pattern Component

Arranging assembly components in a pattern saves time, increases your productivity, and captures design intent. For example, you may need to place multiple bolts to fasten one component to another or place multiple subassemblies into a complex assembly.

You can create a circular pattern by specifying the number of components and the angle between then. You can create a rectangular pattern by specifying column and row spacing. You can create both circular and rectangular patterns by matching features patterned on a part.

Usually, you pattern components at several points in the assembly design process. After you place a component in an assembly. When you place a component:

- You position it using an existing part feature pattern.
- You select the component and copy it into a pattern.

Individual occurrences are listed in the browser as individual parts. Individual or all occurrences can have visibility turned on or off.

You can arrange components in a circular or arc pattern by specifying the number, angle spacing, and rotation axis or by matching the spacing of features a part.

To pattern by specifying the number, angle spacing, and rotation axis:

1. Click the Pattern Component button then click the Circular tab, if necessary.
2. In the browser or graphics window, select one or more components to pattern.
3. In Circular Placement, click an edge or work axis to indicate the Rotation Axis. Click Flip to change the axis direction, if desired.
4. Enter the count (number of features) for the arc or circle and the angular spacing between features.

Apply assembly constraints to position individual components as needed.

To pattern by matching the spacing of features a part:

1. Click the Pattern Component button then click the Circular tab, if necessary.
2. In the browser or the graphics window, select one or more components to pattern.
3. In Circular Placement, click an occurrence of a feature in a pattern.

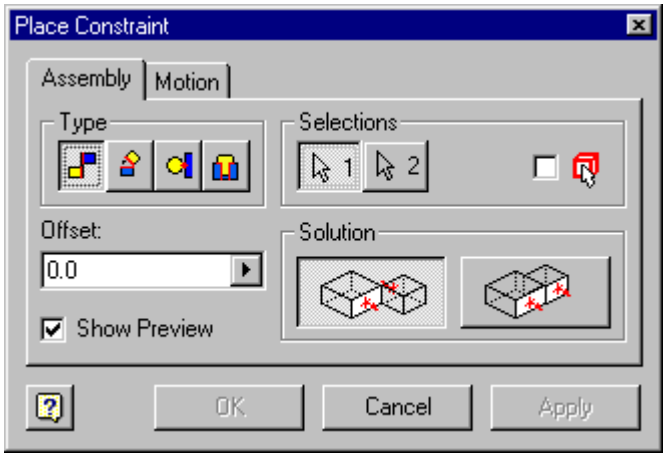
Components are copied in the placement and spacing of the feature pattern. Apply assembly constraints to position individual components as needed.

TIP: In the graphics window, you can select a part feature to pattern, but you must select an occurrence of the feature, not the original feature.

Place Constraint

Assembly constraints determine how components in the assembly fit together. As you apply constraints, you remove degrees of freedom, restricting the ways components can move.

To help you position components correctly, you can preview the effects of a constraint before it is applied. After you select the constraint type, the two components, and set the angle or offset, the components move into the constrained position. You can make adjustments in settings as needed, then apply it.



The Place Constraints dialog box creates constraints to control position and animation. Motion constraints do not affect position constraints.

The Assembly tab has constraints to control position:

- A **mate** constraint positions selected faces normal to one another, with faces coincident or aligns parts adjacent to one another with faces flush. The faces may be offset from one another.
- An **angle** constraint positions linear or planar faces on two components at a specified angle.
- A **tangent** constraint between planes, cylinders, spheres, and cones causes geometry to contact at the point of tangency. Tangency may be inside or outside a curve.
- An **insert** constraint positions cylindrical features with planar faces perpendicular to the cylinder axis.

To create a complex assembly, create several small assemblies and save each one as a separate file. Combine them in larger assemblies, constraining them to other subassemblies and parts as a single unit.

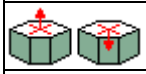
Group parts in subassemblies if you want to use them in more than one assembly. Modify small subassemblies or regroup parts to change assembly configuration.

The four types of assembly constraints are: Mate, Tangent, Angle, and Insert.

Mate Constraint

Mate constraint positions components face-to-face or adjacent to one another with faces flush. Removes one degree of linear translation and two degrees of angular rotation between planar surfaces.	
	Mate constraint positions selected faces normal to one another, with faces coincident.
	Flush constraint aligns components adjacent to one another with faces flush. Positions selected faces, curves, or points so that they are aligned with surface normals pointing in the same direction.

Angle Constraint

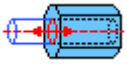
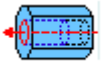
Angle constraint positions edges or planar faces on two components at a specified angle to define a pivot point. Removes one degree of angular rotation.	
	As selected positions components at the specified angle, based on the geometry selected.
	Flip Part 1 rotates the first selected part 180 degrees.
	Flip Part 2 rotates the second selected part 180 degrees.
	Flip both parts rotates both selected parts 180 degrees.

Tangent Constraint

Tangent constraint causes faces, planes, cylinders, spheres, and cones to contact at the point of tangency. Tangency may be inside or outside a curve, depending on the direction of the selected surface normal. A tangent constraint removes one degree of linear translation.	
	Inside Positions the first selected part inside the second selected part at the tangent point.
	Outside Positions the first selected part outside the second selected part at the tangent point. Outside tangency is the default solution.

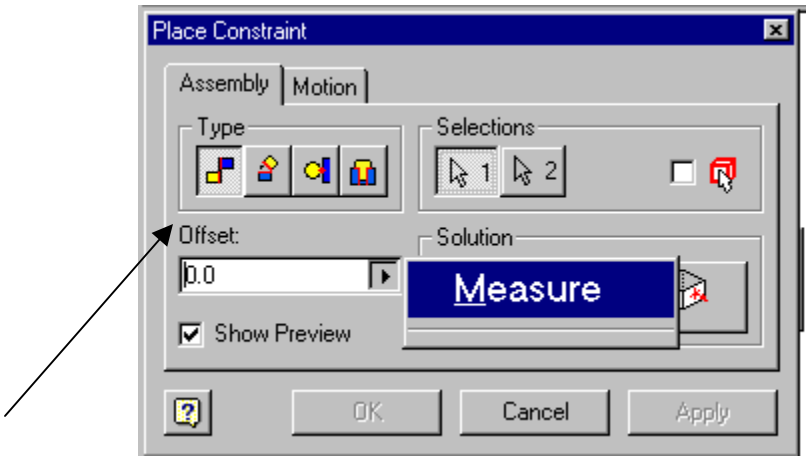
Insert Constraint

Insert constraint is a combination of a face-to-face mate constraint between planar faces and a mate constraint between the axes of the two components. The Insert constraint is used to position a bolt shank in a hole, for example, with the shank aligned with the hole and the bottom of the bolt head mated with the planar face. A rotational degree of freedom remains open.

	Opposed reverses the mate direction of the first selected component.
	Aligned reverses the mate direction of the second selected component.

Selections select geometry on two components to constrain together. You can specify one or more curves, planes, or points to define how features fit together.

	First Selection Selects curves, planes or points on the first component. To end the first selection, click the Second Selection button.
	Second Selection Selects curves, planes, or points on the second component. To select different geometry on the first component, click the First Selection tool and reselect.
	Pick Part First Limits the selectable geometry to a single component. Use when components are in close proximity or partially obscure one another. Clear the check box to restore selection mode.



The Offset text box specifies distance by which constrained components are offset from one another.

Use to enter a value equal to a distance or angle that exists in the assembly, but when you do not know the offset or angle. Click the down arrow to measure the angle or distance between components, show dimensions of selected component, or enter a recently used value.

Specify positive or negative values. Default setting is zero. The first picked component determines the positive direction. Enter a negative number to reverse the offset or angle direction.

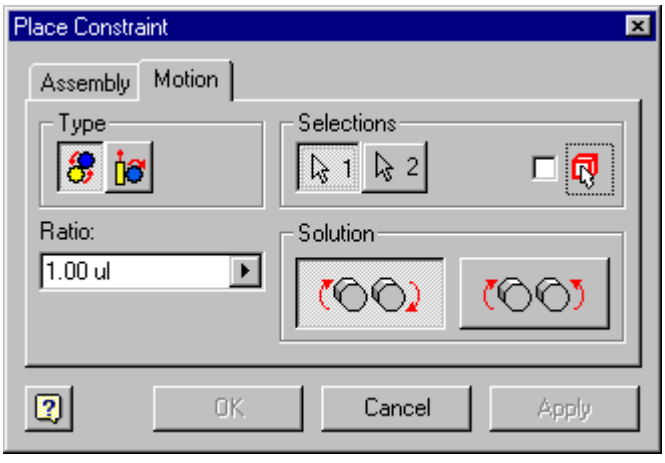
The Show Preview shows the effect of the constraint on the selected geometry. After both selections are made, underconstrained objects automatically move into constrained positions. Default setting is on. Clear the check box to turn preview off.

Motion Constraints

Motion constraints specify the intended motion between assembly components. Because they operate only on open degrees of freedom, they do not conflict with positional constraints, resize adaptive parts, or move grounded components.

Motion constraints are shown in the browser. When clicked or the cursor hovers over the browser entry, constrained components are highlighted in the graphics window.

Drive constraints are not available for motion constraints. However, parts that are constrained using motion constraints will drive according to the direction and ratio specified.



The Motion tab has constraints to specify intended motion ratios between assembly components:

- A **rotation** constraint specifies rotation of one part relative to another part using a specified ratio.
- A **rotation-translation** constraint specifies rotation of one part relative to translation of a second part.

The first part in an assembly is grounded. Its position is fixed, with the part origin coincident with the assembly origin.

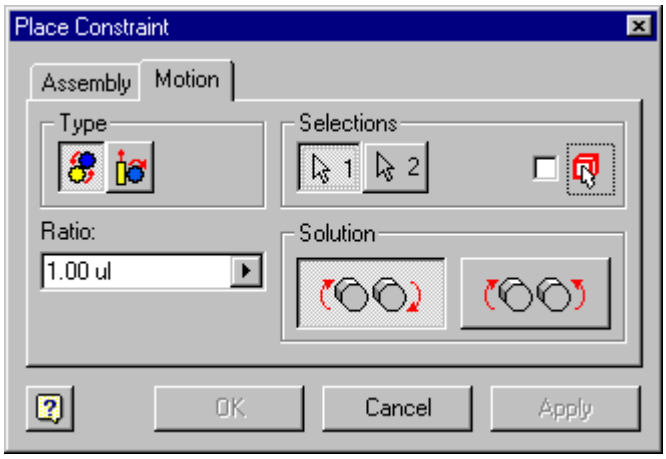
When the next part is placed and constrained to the grounded part, it moves to the grounded part and fits together according to the type of constraint applied.

As you add parts, you can add constraints to position the new parts relative to the other assembled parts.

After constraints are positioned, you can use motion constraints to control rotation and translation in the remaining degrees of freedom. You specify a ratio to set movement between two components.

Drive constraints do not control motion between components, but simulate mechanical motion by driving a constraint through a sequence of steps for a single component. You can, however, animate two components by using the Equation tool to create algebraic relationships between components. A drive constraint operation is a temporary animation.

Motion constraints specify motion ratios between components, either by rotation or by rotation and translation. Such constraints are useful for specifying motion of gears and pulleys, a rack and pinion, or specifying motion between third-party components such as a gearbox and input and output shafts. Use work geometry and assembly constraints to limit the range of motion.



Type specifies the constraint type and illustrates the solution that shows the intended motion between selected components. May be applied between linear, planar, cylindrical, and conical elements. You can change constraint type when the dialog box is open during constraint placement or editing. When the cursor hovers over a component, an arrow shows the direction of the constraint. Click Forward or Reverse to change solution.	
	Rotation constraint specifies that the first selected part rotates in relation to another part using a specified ratio. Typically used for gears and pulleys.
	Rotation-Translation constraint specifies that the first selected part rotates in relation to translation of another part using a specified distance. Typically used to show planar motion, such as a rack and pinions.
Selections select geometry on two components to constrain together. You can specify one or more curves, planes, or points to define how features fit together.	
1	First Selection Selects curves, planes or points on the first component. To end the first selection, click the Second Selection button.
2	Second Selection Selects curves, planes, or points on the second component. To select different geometry on the first component, click the First Selection tool and reselect.
	Pick Part First Limits the selectable geometry to a single component. Use when components are in close proximity or partially obscure one another. Clear the check box to restore selection mode.

Ratio/Distance specifies the movement of the first selected component relative to the second selected component.

Ratio	For Rotation constraints, the ratio specifies how much the second selection rotates when the first selection rotates. For example, a value of 4.0 (4:1) rotates the second selection four units for every unit the first selection rotates. A value of 0.25 (1:4) rotates the second selection one unit for every four units the first selection rotates. The default value is 1.0 (1:1). If two cylindrical surfaces are selected, Autodesk Inventor computes and displays a default ratio that is relative to the radii of the two selections.
Distance	For Rotation-Translation constraints, the distance specifies how much the second selection moves relative to one rotation of the first selection. For example, a value of 4.0 mm moves the second selection 4.0 mm for every complete rotation of the first selection. If the first selection is a cylindrical surface, Autodesk Inventor computes and displays a default distance that is the circumference of the first selection.

TIP:
If you select a Rotation Type, then you must indicate Ratio.

If you select Rotation-Translation Type, then you must indicate Distance.

Although the ratio and distance parameters are used to specify of amount of movement for the second selection with respect to the first selection, the constraint is bi-directional so that if the second selection is moved, the first selection will move by an inverse amount of either the ratio or distance as appropriate to the constraint type.

Replace Component/Replace All

The next tool in the Assembly toolbar is a flyout with two options: Replace Component and Replace All.

Replace Component

In the design process, you often need to replace one or more components in an assembly. You may design a placeholder component that you eventually replace with a standard purchased component, or replace one vendor's component with another.

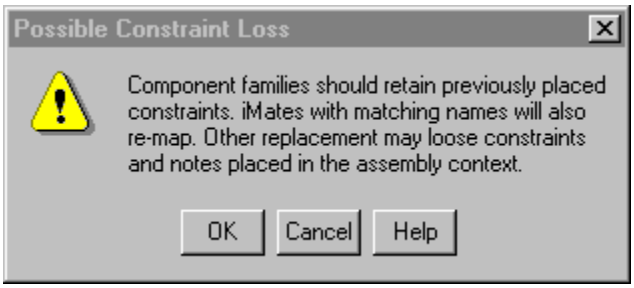
You can select a part to replace an existing assembly component regardless of its location in the directory structure. In a networked environment, you need write permission to replace a component in the open assembly.

The new component is placed in the same location as the original component. The origin of the replacement component is coincident with the origin of the replaced component. Constraints can re-map. Mate and Flush constraints will usually be retained, but Angle and Insert constraints are often lost.

You can replace one assembly component with another component, but existing assembly constraints may be deleted.

1. Click the Replace Component tool, then click a component to replace.

2. In the Open dialog, go to the folder that contains the component, select the component, and click Open.



3. A warning message notifies you that constraints may be deleted. Click OK to continue or Cancel to discontinue replacing a component.

The new component is placed in the same location as the original component. The origin of the replacement component is coincident with the origin of the replaced component. Apply assembly constraints as needed to remove degrees of freedom.

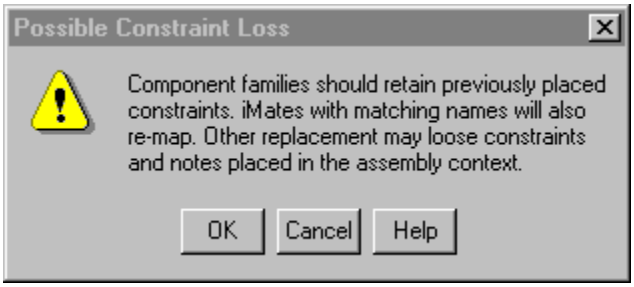
TIP: You can select the component to be replaced either in the browser or by picking a part in the drawing window.

Replace All

The Replace All tool will replace all the occurrences of a component with another component.

1. Click the Replace Component tool, then click a component to replace.

2. In the Open dialog, go to the folder that contains the component, select the component, and click Open.



3. A warning message notifies you that constraints may be deleted. Click OK to continue or Cancel to discontinue replacing a component.

Move Component

When you constrain assembly components to one another, you control their position. To move a component, either temporarily or permanently, use one of these methods:

You can move a component to get a better view of its features. An unconstrained move is simply a temporary "get out of the way" move. You might want to move components to:

- See a face or feature on the selected component.
- See a face or feature on a part that is obscured by the selected component.
- Facilitate selection of a face or feature on a component by moving it to an uncluttered area of the screen.

An unconstrained move is convenient but it is temporary. The part remains in the moved location but snaps back to its constrained position when you apply a new constraint or update or refresh the assembly.

To see how a constrained component moves, you can drag it (and all components constrained to it). A constrained move honors previously applied constraints. That is, the selected component and parts constrained move together in their constrained positions.

A grounded component remains grounded at the new location. Components constrained to the grounded component remain in their constrained positions at the new location.

You can click any component in the edit target (the file that contains edits) and drag it to a new location. If you select a component that is not a child of the edit target (a part in a subassembly), the component acts as a handle and drags the whole subassembly.

You can simulate mechanical motion by driving a constraint through a sequence of steps. After you have constrained a component, you can use the Drive Constraint tool to animate it by incrementally changing the value of the constraint. For example, you can rotate a component by driving an angular constraint from zero to 360 degrees. The Drive Constraint tool is limited to one constraint, but you can drive additional constraints by using the Equations tool to create algebraic relationships between constraints.

Rotate Component

The Rotate tool on the Standard toolbar rotates the entire assembly. When you want to rotate a single component, use the Rotate Component tool on the Assembly toolbar. Operation of both Rotate tools is the same.

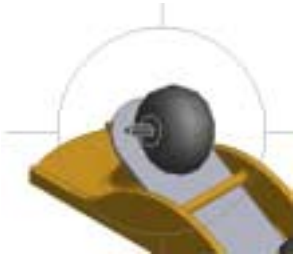
Keep the following behaviors in mind when you rotate components:

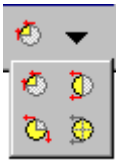
- You can rotate a constrained component.
- When you update the assembly, the components constrained to the rotated component snap into their constrained positions. The rotated component determines the viewing orientation of the components.
- Unconstrained components remain in their rotated positions when you update the assembly.
- Rotating a grounded component overrides the grounded position. The component is still grounded but its position is rotated.

If you require precise positioning of grounded components or other constrained components, use an assembly constraint to reorient a rotated component to the correct position relative to other components.

1. Click the component to rotate.
2. Drag to the desired view of the component.
    - For free rotation, click inside the 3D rotate symbol and drag in the desired direction.
    - To rotate about the horizontal axis, click the top or bottom handle of the 3D rotate symbol and drag vertically.
    - To rotate about the vertical axis, click the left or right handle of the 3D rotate symbol and drag horizontally.
    - To rotate planar to the screen, hover over the rim until the symbol changes to a circle, click the rim, and drag in a circular direction.
    - To change the center of rotation, click inside or outside the rim to set the new center.
3. Release mouse button to drop component in rotated position.

When you click Update, a constrained component snaps back to its constrained position. An unconstrained or grounded component relocates to the new position. Any components constrained to a grounded component snap into their constrained positions in the new location.



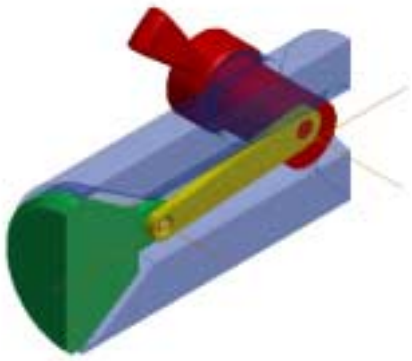


Section Views of Assemblies

You create a section view to visualize portions of an assembly within chambers or that are obscured by components. While the assembly is sectioned, you use part and assembly tools to create or modify parts-in-place.

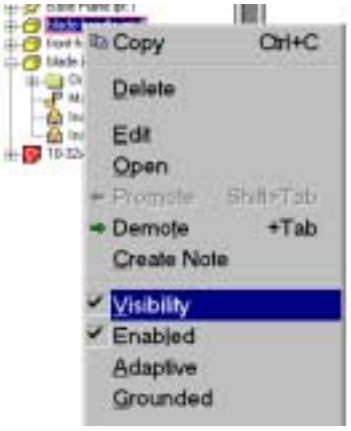
To begin, open an assembly file containing one or more components.

1. Set visibility for components. Select the component in the graphics window or the browser, then:
    - To hide components, right-click and clear the check mark beside Visibility.
    - To show components in wireframe for context, right-click and clear the check mark beside Enabled.
2. Click one of the Section View tools on the Assembly toolbar, then select any planar or work plane to define the cutting plane.
3. Right-click and select Flip, if necessary to display the desired view of the section.
4. Click the Create Component tool on the Assembly toolbar. When prompted to select a sketch plane, select the plane used to define the section.
5. If desired, use the Project Cut Edges tool on the Sketch toolbar to project edges of a part cut by the section plane onto the sketch plane.
6. Use sketch and feature tools to create new geometry.



TIP: From Quarter section and Three-quarter section views, you can right-click and select the opposite view.

Component Visibility

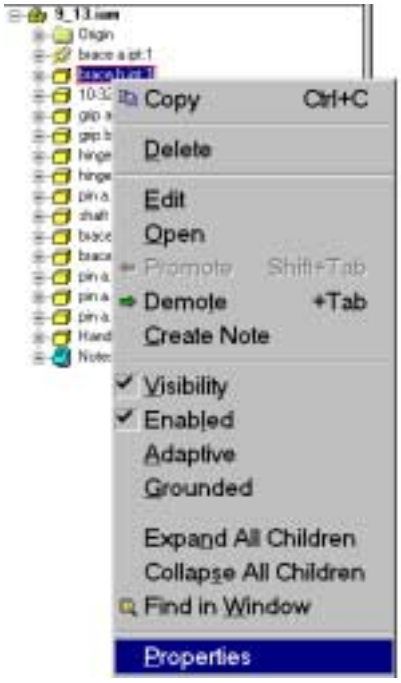


Visible and enabled components can be selected.
Visible but not enabled components cannot be selected. They are displayed in background style (wireframe).
Invisible and not enabled components cannot be selected and are not visible in the graphics window. Select an invisible and disabled components from the browser, then right-click to change its visibility status.
Invisible and enabled components can be selected in the browser but are not visible in the graphics window. The icon shown in the browser looks the same as that for components that are visible and not enabled. Select an invisible and enabled component from the browser, then right-click to change its status.

TIP: It is possible to turn off component visibility, but have the component still be enabled. This may be useful for quickly removing a needed component from view. Enabled components are fully loaded in an assembly file, while only the graphic portion of not enabled components are loaded. The assembly calculates faster because the data structure of not enabled components is not present, but its graphics are useful for a frame of reference.

Occurrence Properties

Occurrence properties control characteristics for an individual occurrence of a component in an assembly.



Click the occurrence in the browser, then right-click and select Properties. Click the Occurrence tab.

1. If desired, enter a descriptive name to replace the default name.
2. Select Visible to make the occurrence visible in the graphics window. Clear the check mark to make it invisible.
3. Select Enabled to make the occurrence selectable in the graphics window. Clear the check mark to change it to Not Enabled.
4. Select Adaptive to allow features in the occurrence to change shape and size when you apply constraints. Clear the check mark to make the occurrence a rigid body.

TIP: The adaptive status of an occurrence controls all occurrences in the assembly. When an adaptive part resizes, all occurrences of the part in other assemblies also resize.

5. Select Grounded to remove all degrees of freedom from the occurrence. Clear the check mark to to restore degrees of freedom.

6. In Color Style, click the down arrow and select from the list.

7. If desired, specify the precise position of the occurrence origin relative to the assembly origin. Enter an offset value or an angle for *x*, *y*, and *z* coordinates.

   Position values are temporary if the occurrence is not grounded. The values are reset when an ungrounded occurrence is moved, rotated, or constrained to a grounded component.

TIP: Except for the current Offset from Parent Assembly Origin, Color Style, or Name, you can set occurrence properties from the context menu. In the browser, right-click the occurrence to view the context menu. A check mark beside the property indicates it is On. Clear the check mark to switch the property Off.

Assembly Tools

Button	Tool	Function
	Place Component	Places a link to an existing part of subassembly in an assembly. A change to any instance updates all other instances of a component.
	Create Component	Creates a new part or subassembly in an assembly
	Pattern Component	Creates copies of a component in a rectangular or circular pattern
	Place Constraint	Places an assembly constraint between two parts.
	Replace Component	Replaces a component in an assembly with another component
	Replace All	Replaces all occurrences of a component in an assembly
	Move Component	Enables a temporary translation of a constrained component. A constrained component returns to proper position when the user clicks Update. Enables permanent translation of a grounded component. A grounded component will remain in the placed position when Update is clicked.
	Rotate Component	Enables a temporary rotation of a constrained component. A constrained component returns to proper position when the user clicks Update. Enables permanent rotation of a grounded component. A grounded component will remain in the placed position when Update is clicked.
	Section Views	Displays a quarter section view of a model defined by hiding portions of components on one side of a defined cutting edge
		Displays a three quarter section view
		Displays a half section view
		Displays an unsectioned view of the model