Autodesk Inventor Tutorial 3

Assembly Modeling

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Assembly Modeling Concepts

With the exception of very simple objects, such as a ruler, most objects have more than one part put together to form a useful whole. The set of parts put together is called an assembly. When you design the parts for an assembly, the relative dimensions and positions of parts, and how they fit together, are crucial. You need to know whether there is any interference among the mating parts. If there is interference, you need to find out where it occurs; then you can eliminate it. Apart from fitting the parts together, you also need to validate relative motions and check clearances if there are moving parts in the assembly. Moreover, you should critically evaluate the parts and the assembly as a whole to ensure the assembly functions correctly in accordance with the design intent. To shorten the design lead time, you construct virtual assemblies in the computer to validate the integrity of the parts and the assembly.

Components

For complex devices that have many parts, it is common practice to organize the parts into a number of smaller sub-assemblies such that each sub-assembly has fewer parts. Therefore, an assembly set consists of an assembly file and a number of part files or an assembly file together with a number of sub-assembly files and part files. Collectively calling the individual parts or sub-assemblies as components, you can define an assembly in the computer as a data set containing information about a collection of components linked to the assembly and how the components are assembled together. Figure 3–1 shows an assembly of a model car. It is a set of components.

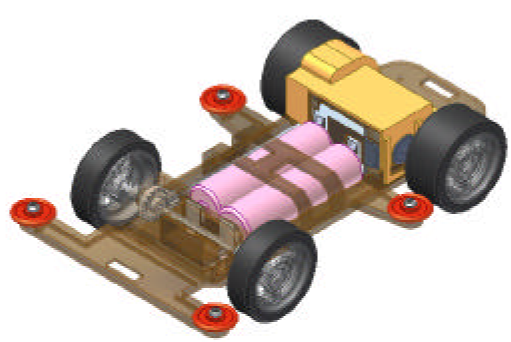

Figure 3–1
An assembly of components

Part File and Assembly File

To construct an assembly in the computer, you start a new assembly file and connect a set of relevant part files and/or assembly files (sub-assemblies) to the assembly file. In the computer, the part files store the information about the 3D objects and the assembly files store the information about how you assemble the 3D objects together. (See Table 6–1.) Because the part files hold the definitions of the solid parts and link to the assembly file, any change you make to the part files will be incorporated in the assembly automatically.

Table 6–1 Information

File	Information
Part File	Definition and information of individual 3D solid part
Assembly File	Information on the locations of the linked components and how the linked components are assembled together

Construction of an assembly

Construction of an assembly in the computer involves two major tasks: gathering a set of components in an assembly file and assembling the components by applying appropriate assembly constraints. How you gather a set of components in an assembly depends on which design approach you take and how you apply assembly constraints depends on the shapes of the features of the components in the assembly.

Design Approaches

There are three design approaches: bottom-up, top-down, (bottom being the parts, and top being the assembly) and hybrid. In a bottom-up approach, you construct all the component parts and then assemble them in an assembly file. In a top-down approach, you start an assembly file and construct the individual component parts while you are doing the assembly. The hybrid approach is a combination of the bottom-up and top-down approaches.

The Bottom-Up Approach

When you already have a good idea on the size and shape of the components of an assembly or you are working as a team on an assembly, you use the bottom-up approach. Through parametric solid modeling methods, you construct all the parts to appropriate sizes and shapes that best describe the components of the assembly. Then you start an assembly file and place the parts in the assembly. In the assembly, you align the components together by applying assembly constraints. After putting all the parts together, you analyze and make necessary changes to the parts.

The Top-Down Approach

Sometimes you have a concept in your mind, but you do not have any concrete ideas about the component parts. You use the top-down approach—you start an assembly file and design some component parts there. From the preliminary component parts, you improvise. The main advantages in using this approach are that you see other parts while working on an individual part and you can continuously switch from designing one part to another.

The Hybrid Approach

In reality, you seldom use one approach alone. You use the bottom-up approach for standard component parts and new parts that you already know about and you use the top-down approach to figure out new component parts with reference to the other component parts. This combined approach is the hybrid approach.

Part Modeling Mode and Assembly Modeling Mode

On matter which design approach you use to construct an assembly, you can switch from assembly modeling mode to part modeling mode (or vice versa) any time you feel appropriate. In assembly modeling mode, you use the assembly modeling tools to construct an assembly. (You will learn more about assembly modeling in this and the next chapter.) In part modeling mode while working in an assembly, the assembly model tools are replaced by the set of part modeling tool and the other component parts in the assembly are grayed out.

Components in an Assembly

In an assembly file, the components are free to translate in three linear directions and three rotational directions. You move and rotate them as if you were manipulating a real object. To impose restriction to the movements and to align a component properly with another component in the assembly, you apply assembly constraints.

Degrees of Freedom

Initially, the component parts (except the first component) that you place or create in an assembly are free to translate in the 3D space, in three linear directions and three rotational directions. These free translations are called the six degrees of freedom (DOF). The DOF of a component is represented by a DOF symbol (see Figure 3–2). You discover the DOF of the objects in an assembly by selecting Degrees of Freedom from the View menu.

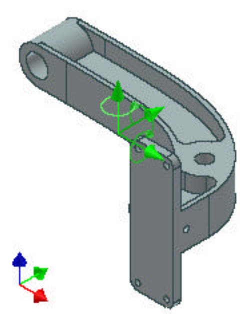

Figure 3–2
Six degrees of freedom of a free object

Grounding

By default, the first component part that you place or create in the assembly is fixed in the 3D space. We call a fixed object a grounded object, it has no degree of freedom—you cannot move it. To free a grounded object, you select the object, right-click, and deselect Grounded. (See Figure 3–3.) On the other hand, if you want to fix an object in 3D space, you ground the object by right-clicking and selecting Grounded.

Translation of Objects in 3D Space

Unless a component part is grounded, it is free to translate in the 3D space. To translate the component parts in an assembly to their appropriate locations, you move or rotate them. Note that moving or rotating a component does not affect a component's DOF. You simply put it in a new position and new orientation.

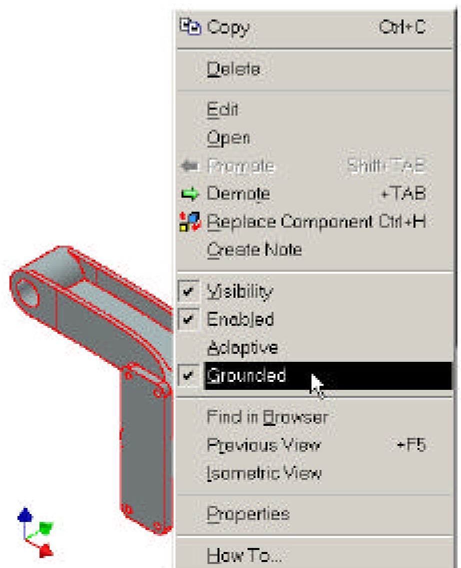

Figure 3–3
Grounded selected

Applying Assembly Constraints

To restrict the movement of a component in 3D and to align it with another component in the assembly, you apply assembly constraints to selected faces, edges, and vertices of parts and the origin (with three work axes along the X, Y, and Z directions) of the part file or the sub-assembly file. There are four kinds of assembly constraints: mate, angle, tangent, and insert. The kind of constraint to be applied to a pair of components depends on the design intend of the assembly and the function and shape of the individual components.

Mate Constraint

A mate constraint causes two selected objects (face, edge, or vertex) to mate or flush at a specified offset distance. You can mate a face to a face, an edge to an edge, a point to a point, an edge to a face, a vertex to a face, and a vertex to an edge. If you mate a face to a face, you have to decide how the faces face each other, in the same direction or in the opposite directions.

By mating a face of a component with six DOFs to a face of another component, you remove three DOFs (two degrees of rotation freedom and one degree of linear freedom).

By mating an edge of a component with six DOFs to an edge of another component, you remove four DOFs (two degrees of rotation freedom and two degrees of linear freedom).

By mating a vertex of a component with six DOFs to a vertex of another component, you remove three DOFs (three degrees of linear freedom).

By mating an edge of a component with six DOFs to a face of another component, you remove two DOFs (one degree of rotation freedom and one degree of linear freedom).

By mating a vertex of a component with six DOFs to a face of another component, you remove one DOF (one degree of linear freedom).

By mating a vertex of a component with six DOFs to an edge of another component, you remove two DOFs (two degrees of linear freedom).

Figure 3–4 shows (from top to bottom) mating face A to face B, mating edge A to edge B, mating vertex A to vertex B, mating edge A to face B, mating vertex A to face B, and mating vertex A to edge B.

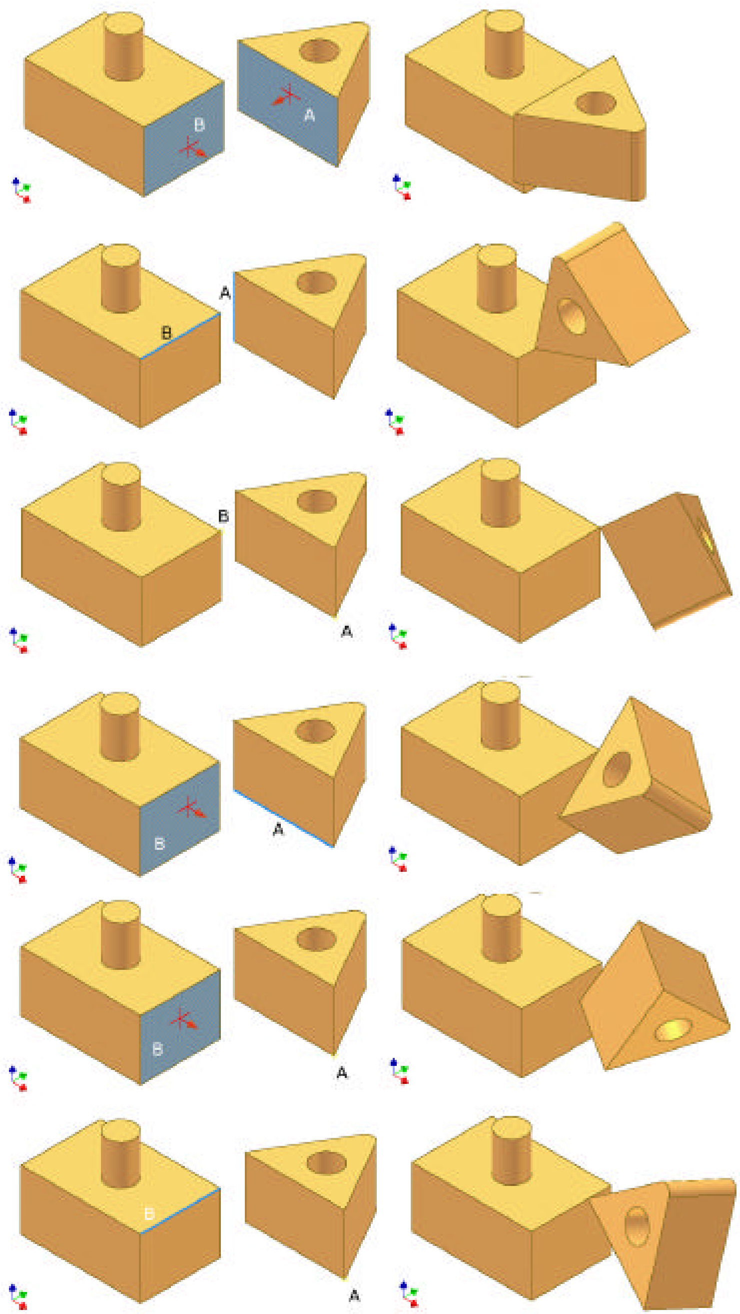

Figure 3–4
Mate constraints

Angle Constraint

Angle constraint causes two selected objects (face or edge) to align at a specified angle. You can constrain a face at an angle with another face, an edge at an angle with another edge, and a face at an angle with an edge. You can consider an angle constraint a special kind of mate constraint because the angle constraint causes the constrained component to align at an angle.

By constraining a face of a component with six DOFs to a face of another component at an angle, you remove three DOFs (two degrees of rotation freedom and one degree of linear freedom).

By constraining an edge of a component with six DOFs to an edge of another component at an angle, you remove four DOFs (two degrees of rotation freedom and two degrees of linear freedom).

By constraining an edge of a component with six DOFs to a face of another component at an angle, you remove two DOFs (one degree of rotation freedom and one degree of linear freedom).

Figure 3–5 shows (from top to bottom) face A constrained at an angle with face B, edge A constrained at an angle with edge B, and edge A constrained at an angle with face B.

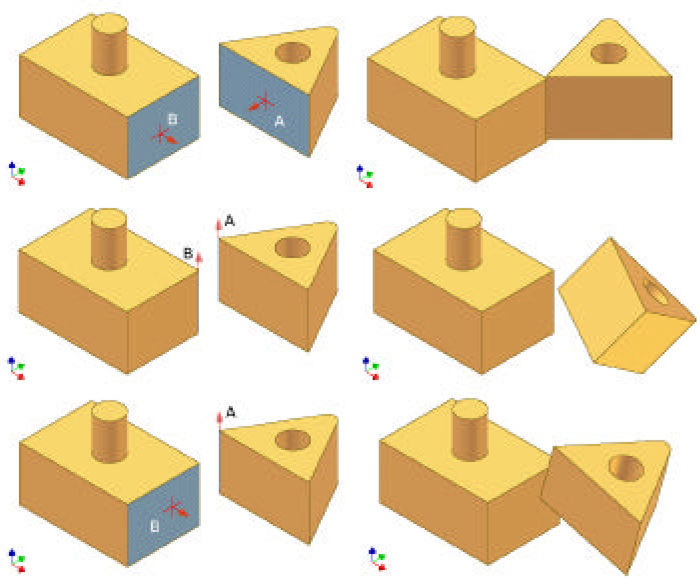

Figure 3–5
Angle constraints

Tangent Constraint

Tangent causes selected faces, planes, cylinders, spheres, and cones to contact at their tangential point and at a specified offset distance; you select faces.

By applying tangent constraint to a cylindrical face of a component with six DOFs to a face of another component, you remove two DOFs (one degree of rotation freedom and one degree of linear freedom).

By applying tangent constraint to a spherical face of a component with six DOFs to a face of another component, you remove one DOFs (one degree of linear freedom).

Figure 3–6 shows (from top to bottom) cylindrical face constrained to tangent with a planar face and a spherical face constrained to tangent with another spherical face.

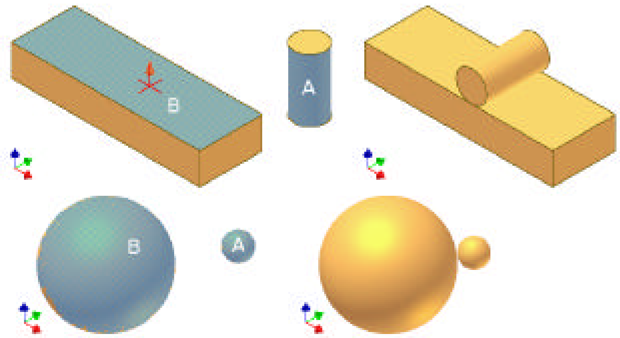

Figure 3–6
Tangent constraints

Insert Constraint

Insert causes selected circular edges to align face to face and concentrically at a specified distance; you select circular edges. An insert constraint is a combination of two mate constraints: mating an axis of a cylindrical object with the axis of another cylindrical object and mating the end face of the cylindrical object with the end face of another cylindrical object.

Figure 3–7 shows (from top to bottom) a cylindrical object inserted to another cylindrical object in an opposed direction and a cylindrical object inserted to another cylindrical object in an aligned direction.

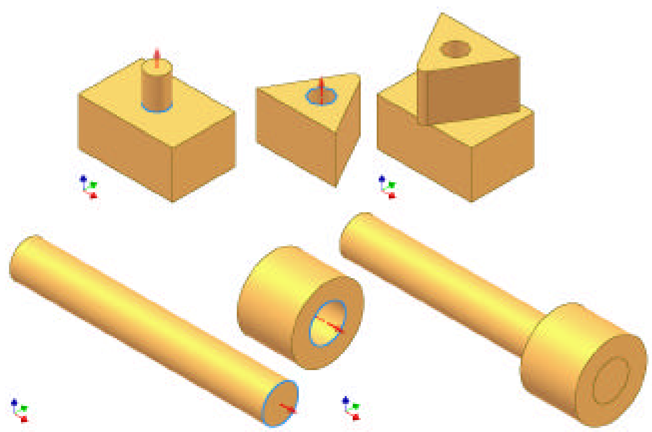

Figure 3–7
Insert constraints