

Autodesk Inventor Tutorial 2

Introduction to Part Modeling

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PARAMETRIC FEATURE-BASED SOLID MODELING CONCEPTS

Autodesk Inventor is a parametric feature-based solid modeling application. Parameters of all objects are modifiable. To construct a 3D solid model, you think about how to decompose it into features, construct the features one by one, and combine them as you construct them. To construct the solid features, you either make sketches or select a pre-constructed solid feature from the menu.

Sketched Solid Features

Sketched solid features derive from sketches. There are four basic kinds of sketched solid features: extruded, revolved, loft, and sweep. In addition, there are other kinds of sketched features: coil, rib, and split.

Extruded Feature

To make an extruded solid feature, you make a sketch and extrude the sketch in a direction perpendicular to the plane of the sketch. (See Figure 2–1.) You extrude the sketch in either direction or from mid-plane.

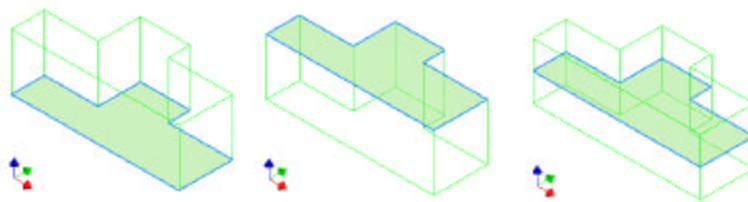


Figure 2–1
Sketch extruded in either direction and from mid-plane

Revolved Feature

To make a revolved solid feature, you make a sketch and revolve it about an axis in either direction or from mid-plane. (See Figure 2–2.)

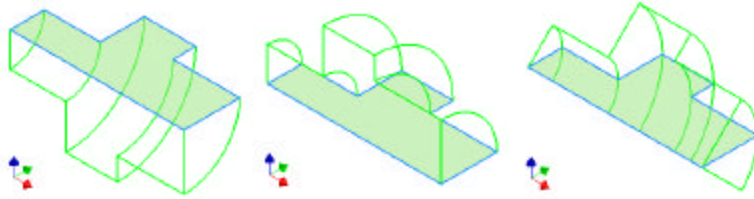


Figure 2-2
Sketch revolved in either direction and from mid-plane

Loft Feature

A loft solid feature builds on multiple sketches. It has a variable cross section defined by two or more sketches residing on different sketch planes. To construct a loft solid, you construct two or more sketches on the different sketch planes, and loft from one sketch to another sketch. Figure 2-3 shows a set of sketches and a loft solid constructed from the sketches.

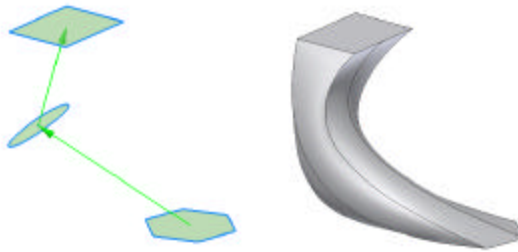


Figure 2-3
Sketches constructed on three sketch planes and loft solid feature constructed from the sketches

Sweep Feature

Construction of a sweep solid feature requires two sketches: a profile sketch and a path sketch. The sweep solid's volume is defined by sweeping the profile sketch along the path sketch. There are two kinds of path sketch: 2D path sketch and 3D path sketch. Sweeping a profile sketch along a 2D path sketch, you construct a 2D sweep solid feature. Sweeping a profile sketch along a 3D path sketch, you get a 3D sweep solid feature.

2D Sweeping

In a 2D sweep feature, both the profile sketch and the path sketch are 2D. They lie on planar sketch planes. Figure 2-4 shows a 2D profile and a 2D path (left) and a 2D sweep solid derived from the sketches (right).

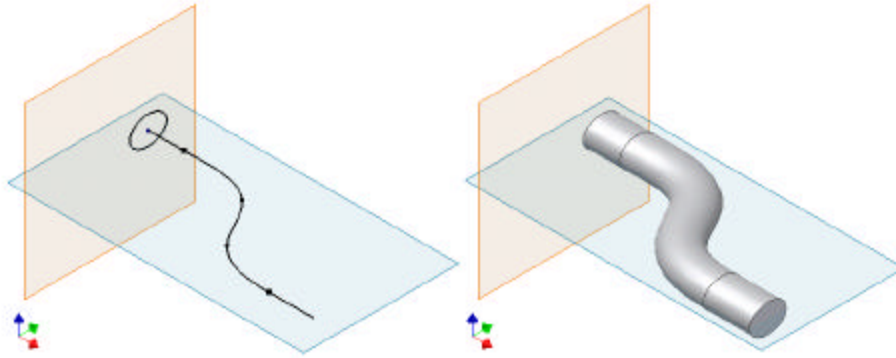


Figure 2-4
Profile sketch and 2D path sketch (left) and 2D sweep solid (right)

3D Sweeping

Construction of 2D and 3D sweep solids is similar, both require a profile sketch to depict the cross section of the sweep solid and a path sketch to specify the path of the sweep operation. The difference between them lies on the shape of the sweep path. In a 2D sweep solid feature, the path sketch is planar. In a 3D sweep solid feature, the path is three dimensional. Figure 2-5 shows a profile sketch and 3D path sketch together with a 3D sweep solid.

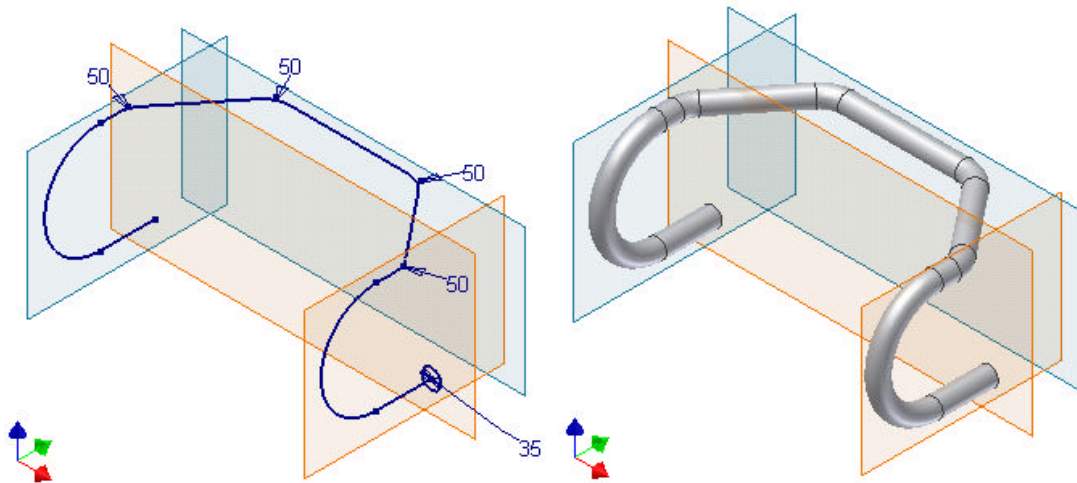


Figure 2-5
Profile sketch and 3D path sketch (left) and 3D sweep solid (right)

Coil Feature

A coil solid is a special kind of 3D sweep solid in which the profile sketch is swept along a helical path. To make a coil solid, you construct a 2D sketch to depict the cross section of the coil, specify an axis, and specify the parameters of the helix. The axis can be a line, an edge, or a work axis. (See Figure 2-6.)

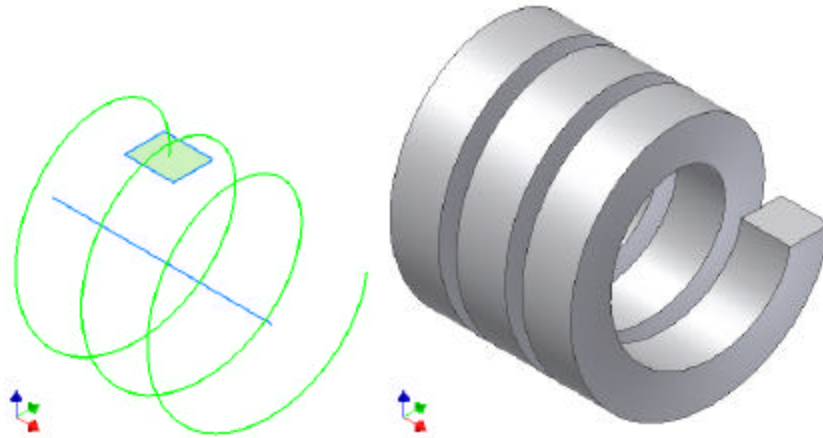


Figure 2-6
Sketch and coil feature

Rib Feature

A rib is a triangular or rectangular reinforcing element you add to a component to strengthen it. (See Figure 2-7.) In essence, a rib is a special kind of extruded solid feature. To construct a rib, you construct an open-loop sketch profile and use the body of the solid to form a close loop in extruding.

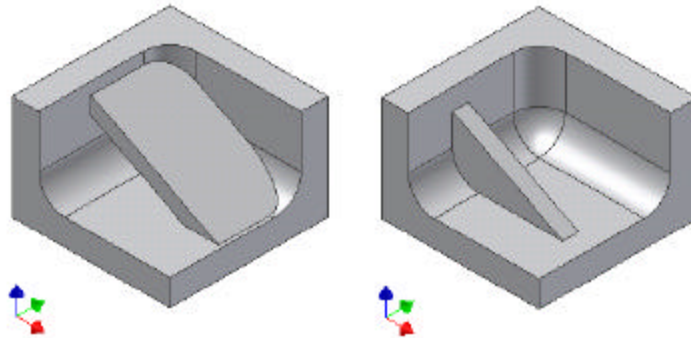


Figure 2-7
Two kinds of rib

Split Features

There are two kinds of split features: face split and part split. Using a split tool, a face split feature splits the faces of a solid part into two pieces, and a part split feature splits a solid part into two solids and removes one of them. You can use a work plane, a sketch, or a surface as split tool. Figure 2-8 shows a face split feature and Figure 2-9 shows a part split feature. Both of them use a sketch as a split tool.

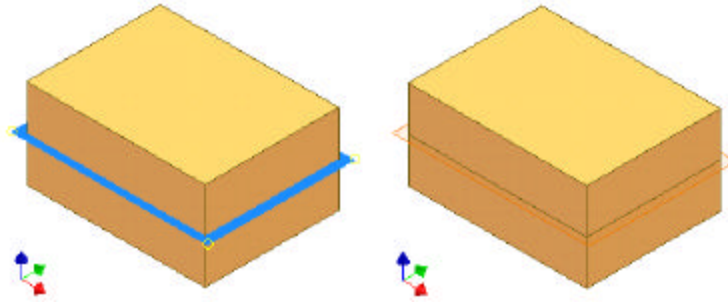


Figure 2–8
Work plane used as split tool (left) and faces split (right)

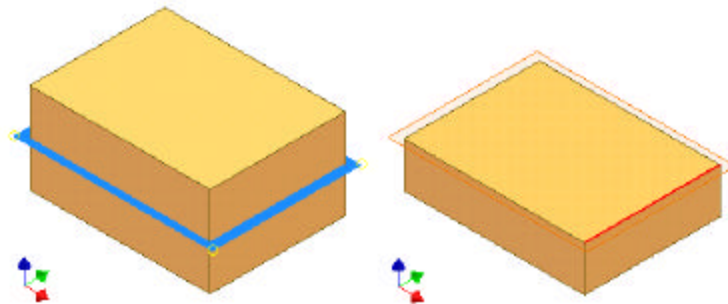


Figure 2–9
Work plane used as split tool (left) and part split (right)

Sketched Surface Features

Using the four basic sketched feature construction methods (extruding a sketch, revolving a sketch, lofting two or more sketches, and sweeping a sketch along another sketch), you can construct surfaces as well as solids. The surfaces that you construct are: extruded surface, revolved surface, loft surface, and sweep surface. To construct surfaces, you select the Surface button from the feature construction dialog box. You can use surfaces as split tool to split a part and as termination objects when constructing an extruded or revolved solid feature.

Sketch Planes

To construct a sketched solid feature, you start with a sketch. To construct a sketch, you need a sketch plane. There are three default planes where you specify a sketch plane: XY plane, XZ plane, and YZ plane. In addition, you can use any existing planes of a solid feature or use artificial planes established by using work features. Figure 2–10 shows a sketch constructed on a face of a solid part.

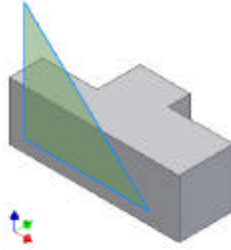


Figure 2–10
Sketch plane established on the face of a solid

Boolean Operations

The first sketched solid feature you construct in a 3D solid part is called the base solid feature. Using feature-based approach to construct a complex solid part, you construct additional features and combine them, one by one, with the existing solid by using one of the three kinds of Boolean operations: join, cut, or intersect.

Join

Using the join operation, you join the new sketched solid feature with the existing solid. The resulting solid has the volume enclosed by the new solid feature and the existing solid. Figure 2–11 shows a revolved solid feature joined to an extruded solid.

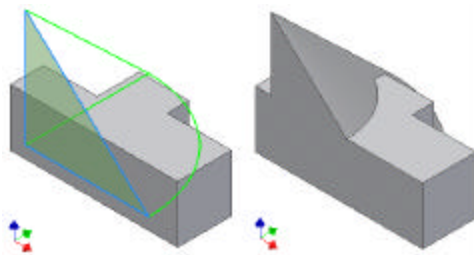


Figure 2–11
New sketched solid feature joined to the existing solid

Cut

Using the cut operation, you cut the new sketched solid feature from the existing solid. The resulting solid has the volume of the new sketched solid feature removed from the existing solid. Figure 2–12 shows a revolved solid feature cut from an extruded solid.

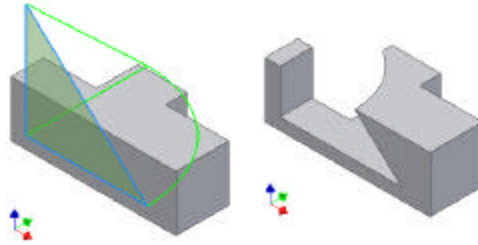


Figure 2–12
New sketched solid feature cut from the existing solid

Intersect

Using the intersect operation, you intersect the new sketched solid feature with the existing solid. The resulting solid has a volume that contains the portion common to both the new sketched solid feature and the existing solid. Figure 2–13 shows a revolved solid feature intersecting with an extruded solid.

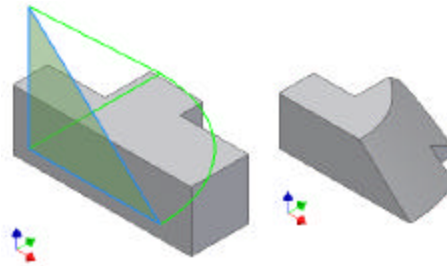


Figure 2–13
New sketched solid feature intersected with the existing solid

Placed Solid Features

Apart from constructing features from sketches, you can construct a feature by selecting a shape from the menu and specify the parameters. We call these pre-constructed solid shapes the placed solid features. Autodesk Inventor has nine kinds of placed solid features: hole, thread, shell, fillet, chamfer, rectangular pattern, circular pattern, mirror, and face draft features.

Hole Feature

A hole is a circular feature cut on a solid. While cutting a hole, you can tap the hole to incorporate an internal thread. To construct a hole, you place a hole feature by specifying type, size, and location. Figure 2–14 shows three hole features placed on a solid.

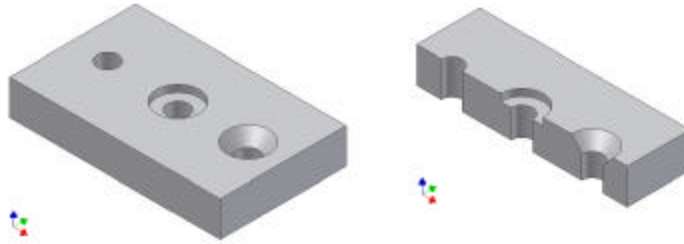


Figure 2-14
An outside view and a cutaway view of a rectangular solid with three hole features

Thread Feature

To construct a thread feature, you select a circular feature and specify the pitch and length of the thread. Figure 2-15 shows two thread features placed on the external and an internal circular features of a solid.

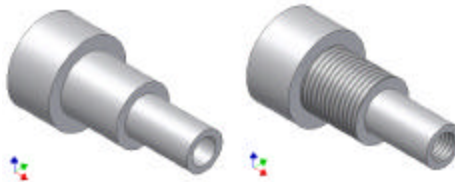


Figure 2-15
Thread features placed on circular faces of a solid part

Shell Feature

To make a solid hollow, you place a shell feature and state the thickness of the shell. Figure 2-16 shows a shell feature placed on a solid part, making the solid hollow.

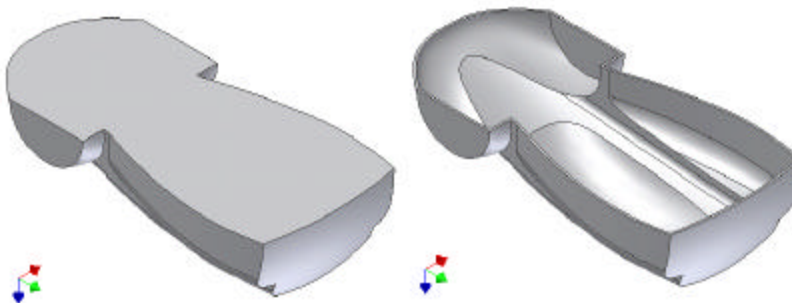


Figure 2-16
Solid part made hollow

Fillet Feature

A fillet feature rounds off the edges of a solid; you select edges and specify fillet radii. Figure 2-17 shows a fillet feature placed on a solid, rounding off three edges and a corner.

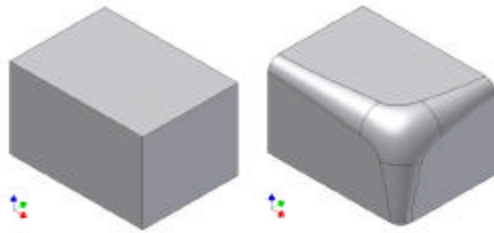


Figure 2–17
Fillet feature placed on a solid

Chamfer Feature

A chamfer feature bevels the edges of a solid; you select edges and specify bevel distances or the bevel distance and bevel angle. Figure 2–18 shows a chamfer feature placed on a solid.

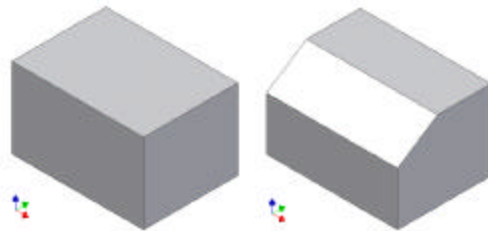


Figure 2–18
Chamfer features placed on a solid

Rectangular Pattern

To repeat a solid feature in a rectangular array, you select the feature and specify the directions, distances, and the number of repetitions. Figure 2–19 shows a rectangular slot repeated in a rectangular array.

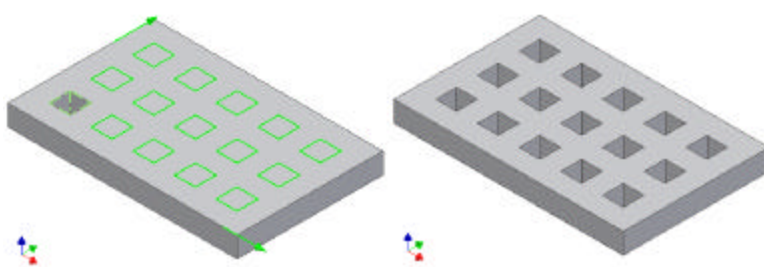


Figure 2–19
Rectangular pattern feature

Circular Pattern

To repeat a solid feature in a circular array, you select the feature and specify an axis, angular distance, and number of repetitions. Figure 2–20 shows a gear tooth repeated in a circular array.

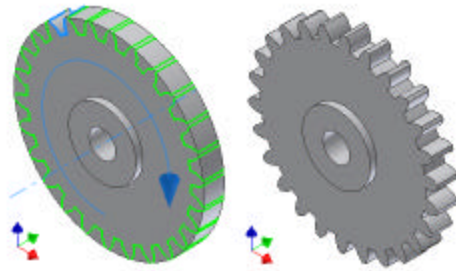


Figure 2–20
Circular pattern feature

Mirror Feature

To construct a mirror copy of a solid feature, you select a feature and specify a mirror plane. Figure 2–21 shows a shaped slot mirrored about an artificial work plane.

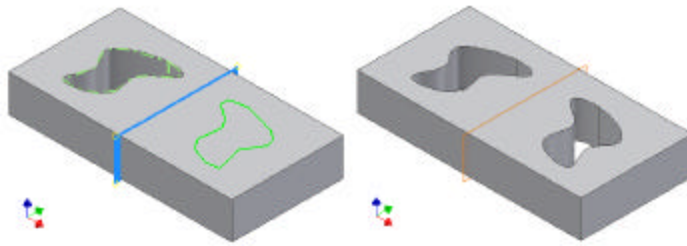


Figure 2–21
Mirror feature placed

Face Draft Feature

To taper the faces of a solid, you place a face draft feature. You select an edge or a split line and specify a draft angle. Figure 2–22 shows a face draft feature placed along the vertical faces of a solid part.

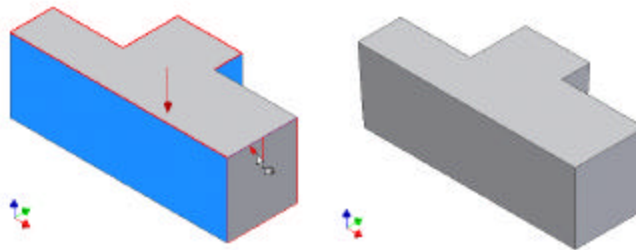


Figure 2–22
Face draft feature placed along the edges of a solid

By now, you should have a general picture of two kinds of sketched solid features (extruded and revolved) and various kinds of placed solid features (hole, thread, shell, fillet, chamfer, rectangular pattern, circular pattern, mirror, and face draft) that you use in making a solid model.

WORK FEATURES

To help establish sketch plane and references for solid modeling, you construct artificial planes, axes, and points. They are work points, work axes, and work planes. Collectively, we call them work features.

Features in a Solid Part

Together with sketched solid features and placed solid features, there can be three kinds of features in a solid part. They are:

1. Sketched Solid Features
2. Placed Solid Features
3. Work Features