Autodesk Inventor 2010

Getting Started

Autodesk

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Digital Prototypes in Autodesk Inventor

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Autodesk Inventor® provides a comprehensive set of 3D mechanical CAD tools for producing, validating, and documenting complete digital prototypes. The Inventor model is a 3D digital prototype. The prototype helps you visualize, simulate, and analyze how a product or part works under real-world conditions before it is built. Manufacturers get to market faster with fewer physical prototypes and more innovative products.

Inventor provides an intuitive 3D design environment for creating parts and assemblies. Engineers can focus on the function of a design to drive the automatic creation of intelligent components, such as steel frames, rotating machinery, tube and pipe runs, electrical cables, and wire harnesses.

Tightly integrated motion simulation and stress analysis in Inventor are easy to use. They make it possible for engineers to optimize and validate the digital prototype.

Generating manufacturing documentation from a validated 3D digital prototype reduces errors and associated engineering change orders (ECOs) before manufacturing. Inventor offers rapid and accurate output of production-ready drawings directly from the 3D model.

Inventor is tightly integrated with Autodesk® data management applications. This integration enables the efficient and secure exchange of digital design data and promotes earlier collaboration between design and manufacturing workgroups. Different workgroups can manage and track all components of a digital prototype with Autodesk® Design Review software. This software is the all-digital way to review, measure, mark up, and track changes to designs. You can better reuse crucial design data, manage bills of materials (BOMs), and collaborate with other teams and partners.

Digital Prototype Workflow

Before you start a design, determine the most efficient workflow. A top down workflow is often the most efficient way to create a design. In a top down

workflow, you design your components in the context of other components. This method can greatly reduce errors in form, fit, and function.

Some examples of a top-down workflow are:

- Create new parts or sub-assemblies in the destination assembly.

- Create multiple solid bodies in a part file and then save the individual bodies as unique parts.

- Create 2D sketch blocks in a part file to simulate a mechanism. You can use the sketch blocks to create 3D components in an assembly that is controlled by the layout.

Following are questions to consider before you start:

- Which view of the part best describes the basic shape?

- Is the part a sheet metal part?

- Can this part be used as a part factory (iPart) to generate multiple parts?

- Can a spreadsheet control one or more parts?

- Can I create the part automatically by using a Design Accelerator?

- If the part is a component in a structural steel frame, can I use Frame Generator to create the entire frame?

- If the part is a common library part, does it exist in the Content Center or other library?

The following image shows a multi-body part file saved as individual parts in an assembly. Individual bodies in a multi-body part file can share features with other bodies such as fillets and holes.
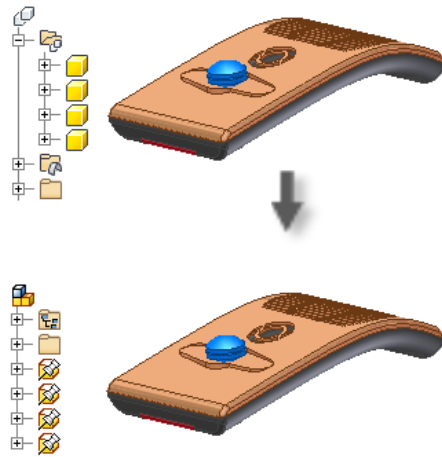
For more information	Location
Help topic	Search: "Multi-body parts"
Tutorial	Parts 1 - Create Parts
Skill Builders	Parts

Components of Digital Prototypes (file types)

Create or activate a project file before you open an existing file or start a new file to set the file location. Click New to see the New File dialog box with templates for a new part, assembly, presentation file, sheet metal part, weldment, or drawing. You can choose from several templates with predefined units.

A template can contain property information, such as part and project data, and drawing views. You can see information stored in a file by viewing its properties.

For more information about templates, see Templates on page 67. For more information about projects, see Projects on page 68.

For more information about projects, see Projects on page 68.

Part (.ipt) Files

When you open a part file, you are in the part environment. Part tools manipulate sketches, features, and bodies which combine to make parts. You can insert a single body part into assemblies and constrain them in positions they occupy when the assembly is manufactured. You can extract multiple part files from a multi-body part.

Most parts start with a sketch. A sketch is the profile of a feature and any geometry (such as a sweep path or axis of rotation) required to create the feature.

A part model is a collection of features. If necessary, solid bodies in a multi-body part file can share features. Sketch constraints control geometric relationships such as parallel and perpendicular. Dimensions control the size. Collectively this method is called Parametric modeling. You can adjust the constraints or dimensional parameters that control the size and shape of a model, and automatically see the effect of your modifications.

The following image shows a single body part in the upper half of the image, and a multi-body part in the lower half of the image. Notice the different part icons in each image.
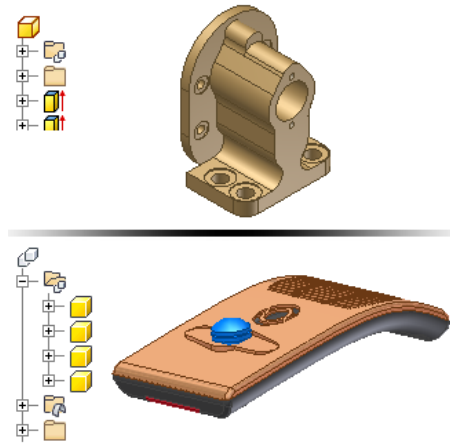
Assembly (.iam) Files

In Autodesk Inventor, you place components that act as a single functional unit into an assembly document. Assembly constraints define the relative position these components occupy with respect to each other. An example is the axis of a shaft aligning with a hole in a different component.

When you create or open an assembly file, you are in the assembly environment. Assembly tools manipulate whole subassemblies and assemblies. You can group parts that function together as a single unit and then insert the subassembly into another assembly.

You can insert parts into an assembly or use sketch and part tools to create parts in the context of an assembly. During these operations, all other components in the assembly are visible.

To complete a model, you can create assembly features that affect multiple components, such as holes that pass through multiple parts. Assembly features often describe specific manufacturing processes such as post-machining.

The assembly browser is a convenient way to activate components you want to edit. Use the browser to edit sketches, features, and constraints, turn component visibility on and off, and do other tasks. In the following image of an assembly, two of the components display an icon indicating they are part of a contact set. Components that belong to a contact set behave as they would in the physical world.
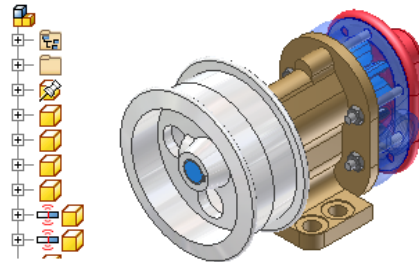
Presentation (.ipn) Files

Presentation files are a multi-purpose file type. Use a presentation file to:

- Create an exploded view of an assembly to use in a drawing file.

- Create an animation which shows the step by step assembly order. The animation can contain view changes and the visibility state of components at each step in the assembly process. You can save the animation to a *.wmv* or *.avi* file format.
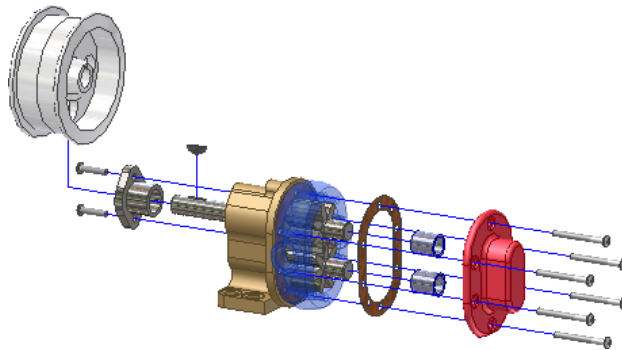
Drawing (.idw, .dwg) Files

After you create a model, you can create a drawing to document your design. In a drawing, you place views of a model on one or more drawing sheets. Then you add dimensions and other drawing annotations to document the model.

A drawing that documents an assembly can contain an automated parts list and item balloons in addition to the required views.

PARTS LIST		
ITEM	QTY	PART NUMBER
1	1	Pump Base
2	4	Gear Bushing
3	2	Gasket
4	1	Ppump Body
5	1	Driven Gear
6	1	Drive Gear
7	1	Gland Bushing
8	1	Gland
9	1	Woodruff Key
10	1	Pump Cover
11	1	Pulley
12	4	10-32 Nut
13	6	10 - 32 - 1 3/4
14	2	10 - 32 - 7/8
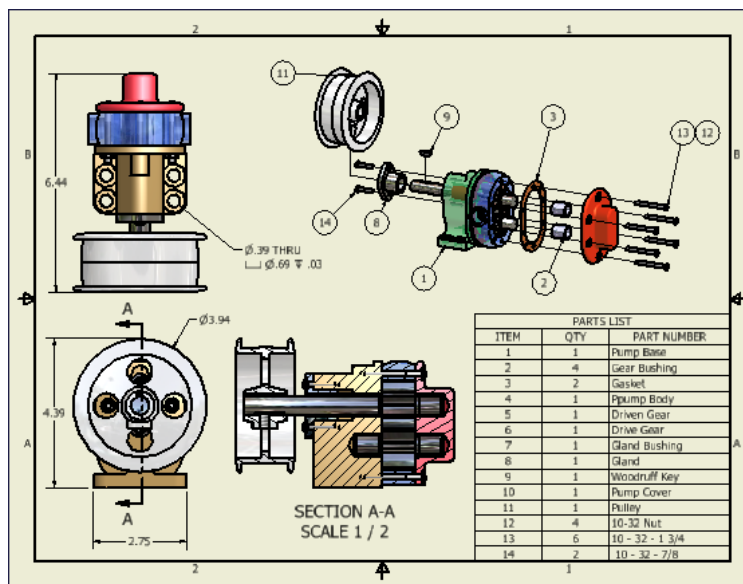
The templates to use as the starting point for your drawings have the standard drawing file extension (*.idw*, *.dwg*).

Autodesk Inventor maintains links between components and drawings, so you can create a drawing at any time during the creation of a component. By default, the drawing updates automatically when you edit the component. However, it is a good idea to wait until a component design is nearly complete before you create a drawing. Edit the drawing details (to add or delete dimensions or views, or to change the locations of notes and balloons) to reflect the revisions.

For more information	Location
Help topic	Search: "Autodesk Inventor file types" "Set file names"
Autodesk Vault Implementation Guide	Manual in PDF format.

Associative Behavior of Parts

Other than the origin work planes, work axes, the center point, and grounded work points, all work features are associated to the features or geometry used to create them. If you modify or delete the locating geometry, the work feature changes accordingly. Conversely, changes to the work feature affect any feature or geometry that is dependent on a work feature for its definition.

A parent-child relationship is a term frequently used to describe the relationship between features. A child feature cannot exist without the parent feature. If you delete a parent feature, you can choose to retain the originating sketch of a child feature. If you create geometry on an origin plane or a work plane created from an origin plane you can often avoid creating parent-child relationships.

A derived part can maintain associative links to the source component so it can be updated. You can also choose to break the link between the derived part and the source part or assembly to disable updates.

For more information about derived parts and work features, see Parts on page 9 and Features on page 18.

Associative Behavior of Assemblies

An assembly maintains active links to the source components. Each time you open an assembly, Inventor detects the latest version of the components

contained in the assembly. When you open an assembly file in which one or more components are modified, a message displays asking if you want to update the assembly. Answer yes to update the assembly to the last saved state of the components. Answer no to disregard any modifications to the referenced components.

Associative Behavior of Drawings

Drawings maintain associativity to the components contained in the file views. If you change a component, the component view automatically updates the next time the drawing file is opened. You can choose to disable automatic updates by enabling Defer Updates in the Drawing tab of Document Settings.

If the drawing contains a parts list and item balloons, the balloon numbers are associative to the item numbers in the parts list. The parts list is also associative to the Bill of Materials in the source assembly. If items are deleted from the assembly, they are no longer contained in the drawing parts list. The parts list is associative to the iProperties of the components being detailed for entries such as part number and description.

For more information	Location
Help topic	Search: "3D modeling concepts" "2D to 3D bidirectional associativity" "Assembly components in patterns" "Design view representations in drawings"

Create Digital Prototypes

2

Traditionally, designers and engineers create a layout, design the parts, and then bring everything together in an assembly. Once the design is created, the next step in the traditional process is to build and test a physical prototype.

NOTE This chapter describes how to create digital prototypes in Inventor LT

With Autodesk Inventor®, you can create an assembly at any point in the design process. You can virtually explore, test, and validate a digital prototype as the design evolves. You can visualize and simulate real-world performance of the design, so there is less reliance on costly physical prototypes.

The basic component of a digital prototype is the part file. A part model is a collection of features or solid bodies that define your digital prototype. Parametric modeling provides the ability to apply driving dimensions and geometric relationships to the model. These dimensions and relationships are called parameters. Parameters control the size and shape of a model. When you change a parameter, the model updates to reflect the changes. Using parameters, you can control multiple parts in an assembly.

Parts

A file with an *.ipt* extension represents a part file. A part is represented on disk with only one file type. However, there are many different types of part files. They can be simple to complex. Some of the common types of parts are explained in the following section. The workflow you use to create the part is what determines the part type.

Single Body Parts

The most basic part type can vary greatly in complexity from just a few features to a complex design. The distinguishing features are that it is composed of one material and one solid body, of which the thickness can vary.
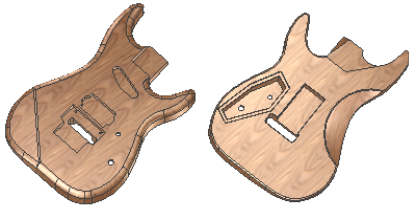
A single body part contains one solid body that shares a collection of one or more features. A single body part defines a single item in a parts list.

For more information	Location
Help topics	Search: "Create parts in assemblies" "Work with parts"
Tutorial	Parts 1 - Create Parts

iParts

Most designers have parts that differ by size, material, or other variables, but the same basic design works in many models.
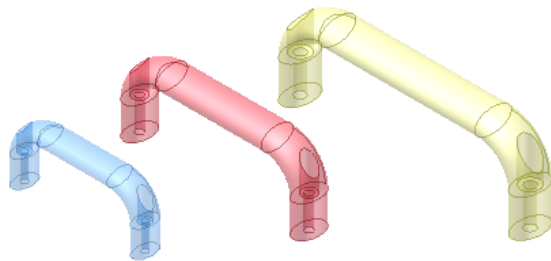
An iPart is a table driven master part that configures standard parts to different sizes and states. The table can be edited within Inventor or externally in a spreadsheet.

Each row can control feature state (enabled or suppressed), and many other variables such as feature size, color, material, and part number. Table driven iFeatures can also be included in an iPart table.

An iPart typically generates multiple unique parts that belong to the same family.

NOTE You can create an iPart and save it as a table-driven iFeature.

Use the iPart Author to create the part family members in each table row. When placing the part in an assembly, select a row (member) to generate a unique part.

For more information	Location
Help topic	Search: "iPart Fundamentals"
Skill Builder	Parts: iParts - The Basics iParts - Beyond the Basics

Sheet Metal Parts
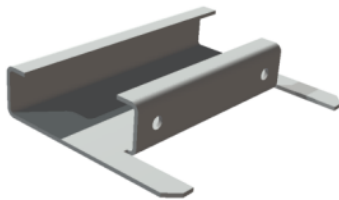
Chances are that a design you have been asked to create contains components that lend themselves to fabrication from sheet metal.
Autodesk Inventor provides functionality that simplifies the creation, editing, and documentation of digital prototypes of sheet metal components.
A sheet metal part is often thought of as a part fabricated from a sheet of uniformly thick material. If you design small objects, this material is often thin. However, in Autodesk Inventor you can utilize the sheet metal commands on any design where the material is of uniform thickness.

Within the Autodesk Inventor design environment, a sheet metal part can be displayed as a folded model or a flat pattern. With sheet metal commands, you can unfold features and work on a model in a flattened state, and then refold the features.

You create sheet metal parts from a template file. The sheet metal template file incorporates a set of rules. The rules determine some common attributes such as material type and thickness, unfolding rules, gap sizes, and so on. By changing a single rule, you can change the material of a sheet metal part from

aluminum to stainless steel. A change of material often requires changes to the attributes that define bends and corners. Such changes often require changes to shop floor machinery and set-ups used to fabricate the parts.

Like other parts created within Autodesk Inventor, sheet metal parts begin with a base feature. The base feature of a sheet metal part is often a single face of some shape to which other features (often flanges) are added. A complex design could use a contour flange or contour roll as the initial base feature. Some parts could utilize a lofted flange as the initial feature.

Unlike regular parts, sheet metal parts are always created from a uniformly thick sheet that is flat. This sheet is formed into the final part using various fabrication techniques. In the sheet metal environment, you can create a folded model and unfold it into a flat pattern. The flat pattern is typically used to detail the fabrication. The sheet metal commands you use to work with flat patterns can provide critical fabrication information.

If a regular part created in Autodesk Inventor is of a consistent thickness, you can convert it to a sheet metal part. The same is true for parts imported from other systems.

For more information	Location
Help topics	Search: "Sheet Metal Defaults" "Templates for sheet metal parts"
Tutorial	Build sheet metal parts
Skill Builders	Parts: Sheet Metal Punch iFeatures - part 1 Sheet Metal Punch iFeatures - part 2

Derived Parts
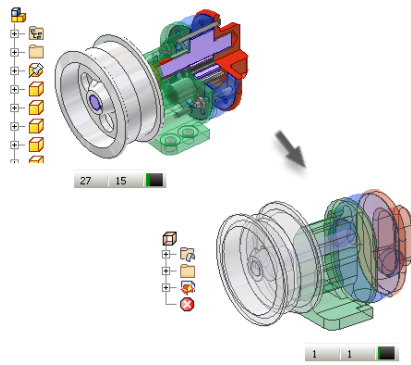
A derived part is a new part or body created from an existing part or assembly.

Use Derived Component to:

■ Create modified or simplified versions of other components.

■ In an empty part file, create a derived part from another part or assembly.

■ In a multi-body part, insert components as tool-bodies.

■ Mirror or scale a part or assembly

■ Perform Boolean operations.

A derived part can contain features independent of the parent component, and can be:

■ Driven by the original component or the link can be disabled.

■ Used for scaling and mirror operations.

■ Derived from a specific assembly Level of Detail.

■ Used to perform add and subtract operations on assembly components.

■ An existing component inserted as a new toolbody in a multi-body part file.

For more information	Location
Help topics	Search: "Derived parts and assemblies" "Derived parts"
Tutorial	Create Parts from Derived Geometry
Skill Builder	Parts - Derived Parts

Multi-body Parts

Multi-body parts are used to control complex curves across multiple parts in plastic part design or organic models.

A multi-body part is a central design composed of features contained in bodies that can be exported as individual part files.

You can insert components into a multi-body part file with the Derived Component command. Use the Combine command to perform Boolean operations.

For more information	Location
Help topic	Search: "Combine solid bodies"
Tutorial	Explore Multi-Bodies and Plastic Features

Shrinkwrap Parts

A shrinkwrap part uses the derived component mechanism to create a simplified part file from an assembly. The Shrinkwrap command uses rule based face and component removal and hole patching to simplify an assembly. A shrinkwrap surface composite (the default setting) uses less memory and

provides the best performance when used as a substitute LOD in consuming assemblies.
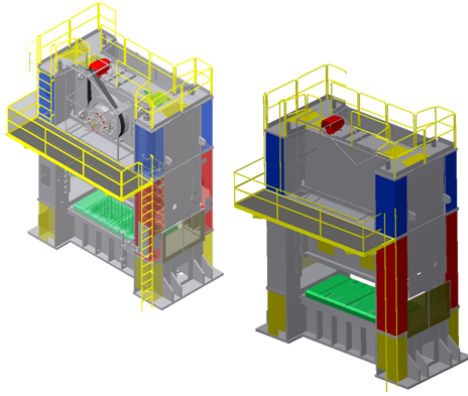
Use Shrinkwrap to:

- Create an envelope of an assembly to provide information to an outside group such as AEC.

- Create a part that uses less memory and provides better performance in consuming assemblies.

- Create a part that protects intellectual property by concealing holes and components.

- Create a simplified part to use as a substitute LOD in the owning assembly.

NOTE A shrinkwrap part is created from an assembly to remove parts and small features from the assembly. Use a shrinkwrap part to simplify a design or protect intellectual property.

For more information	Location
Help topic	Search: "Shrinkwrap assemblies"

Assembly Substitute Parts

An assembly substitute part is a simplified representation of an assembly. It can be created from any part file on disk, or derived in place in the owning assembly. You can create a shrinkwrap substitute part in an assembly to reduce file size and complexity.
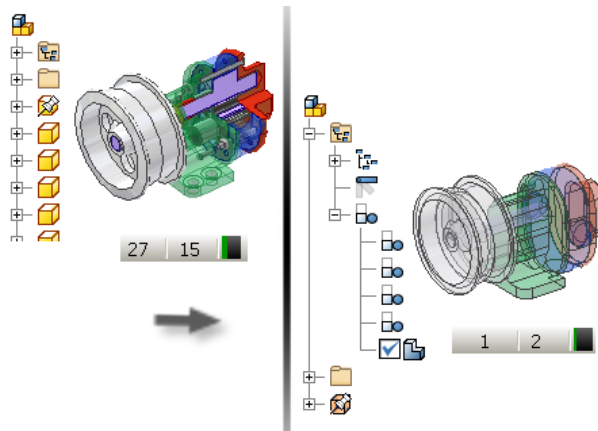
For more information	Location
Help topic	Search: "Create Substitutes"

Content Center Parts

Autodesk Inventor Content Center libraries provide standard parts (fasteners, steel shapes, shaft parts) and features to insert in assemblies.

Two types of parts are included in the Content Center library: standard parts and custom parts. Standard parts (fasteners, shaft parts) have all part parameters defined as exact values in the table of parameters. Custom parts (steel shapes, rivets) have a parameter set arbitrarily within the defined range of values.

The basic component in a Content Center library is a family (part family or feature family). A family contains family members that

have the same template and family properties, and represent size variations of a part or feature.
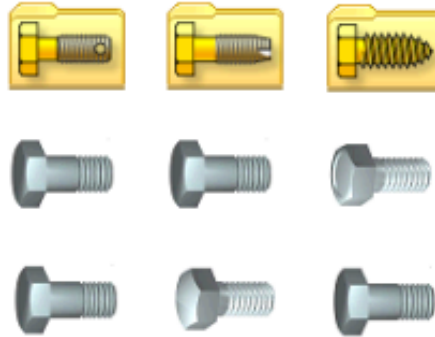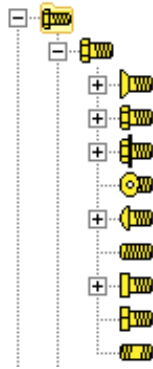
Families are arranged in categories and subcategories. A category is a logical grouping of part types. For example, studs and hex head bolts are functionally related and are nested under the Bolts category. A category can contain subcategories and families.

Use the Content Center environment to work with Content Center library parts in the design process.

- Open and view a part family, and choose the family member.

- Insert a part from Content Center library into an assembly file.

- Insert a feature from Content Center library in a part.

- Use AutoDrop to place a part interactively from a Content Center library into an assembly file.

- Change the size of a placed Content Center library part.

- Replace an existing (also non-Content Center) part with a part from the Content Center library.

For more information	Location
Help topic	Search: "Content Center Environment"
Tutorial	Use Content Center
Skill Builders	Content Center

Content Center Libraries

Content Center libraries contain data required to create part files for Content Center library parts. The data are:

■ Parametric *.ipt* files which provide models for Content Center library parts.

■ Family tables which include values of part parameters.

■ Descriptions for parts including family properties such as family name, description, standard, and standard organization.

■ Preview pictures displayed in the Content Center.

Parametric *.ipt* files, description texts, and preview pictures are common for all members of one family. Sets of parameter values specify particular family members.

A set of standard Content Center libraries can be installed with Autodesk Inventor. Standard libraries are read-only and cannot be edited directly. You must copy parts to the read/write library first.

Use the Content Center Editor to build user libraries and to modify or expand standard content delivered with the Autodesk Inventor installation.

For more information	Location
Help topic	Search: "Content Center Editor"
Tutorial	Content Center User Libraries

Features

The building blocks of a part model are called features. There are four basic types of Features:

■ Sketched Features that require a sketch.

■ Placed Features that modify existing geometry. For example, a hole is a placed feature.

■ Work Features used for construction purposes.

■ iFeatures that represent common shapes and are saved in a reusable library. An iFeature driven by a table can represent different shape configurations.

You can create surfaces with many of these operations to define shapes or aspects of the part body. For example, you can use a curved surface as a termination plane for cuts in a housing.

You can edit the characteristics of a feature by returning to its underlying sketch or changing the values used in feature creation. For example, you can change the length of an extruded feature by entering a new value for the extent of the extrusion. You can also use equations to derive one dimension from another.

For more information	Location
Help topics	Search: "Adaptive features, parts, and subassemblies"
Tutorial	Create and Reuse iFeatures

Sketched Features

Most parts start with a sketch. A sketch is the profile of a feature and any geometry (such as a sweep path or axis of rotation) required to create the feature. Your first sketch for a part can be a simple shape.

Sketched part features depend on sketch geometry. The first feature of a part, the base feature, is typically a sketched feature. All sketch geometry is created and edited in the sketch environment, using Sketch commands on the ribbon. You can control the sketch grid, and use sketch commands to draw lines, splines, circles, ellipses, arcs, rectangles, polygons, or points.

You can select a face on an existing part, and sketch on it. The sketch displays with the Cartesian grid defined. If you want to construct a feature on a curved surface, or at an angle to a surface, first construct a work plane. Then sketch on the work plane.

The browser displays the part icon, with its features nested under it. Surface features and work features are nested or consumed by default. To control nesting, or consumption of surface and work features for all features, set the option on the Part tab of the Applications Options dialog box. To override consumption on a per-feature basis, right-click the feature in the browser, and then select Consume Inputs.

The following features are dependent on a sketch you create:

Extrude

Adds depth to a sketch profile along a straight path.

Can create a body.
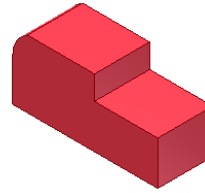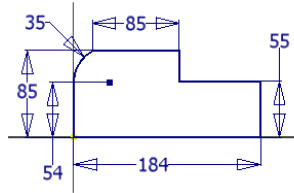
Revolve

Projects a sketch profile around an axis.

The axis and the profile must be co-planar.

Can create a body.
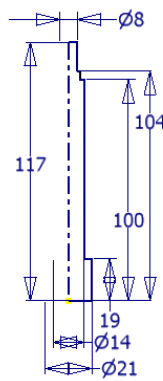
Loft

Constructs features with two or more profiles.
.
Transitions the model from one shape to the next.

Aligns the profiles to one or more paths.

Can create a body.
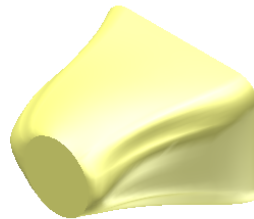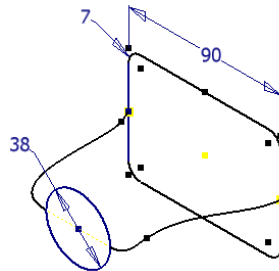
 Sweep

Projects a single sketch profile along a single sketched path.

The path can be open or closed.

A sketch profile can contain multiple loops that reside in the same sketch.

Can create a body.
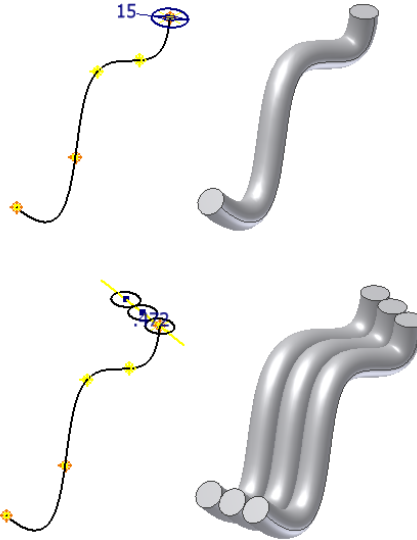
 Coil

Projects a sketch profile along a helical path.

Use Coil to create springs or to model physical threads on a part.

Can create a body.
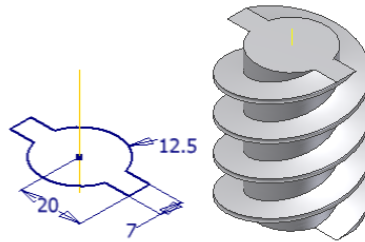
The models created by these operations are typically solid features or new bodies that form a closed volume.

Surfaces

You can create surfaces with many of these operations. Surfaces can form an open or closed volume but contain no mass. Use surfaces to define shapes, use as a split tool, or sculpt certain aspects of the part body.
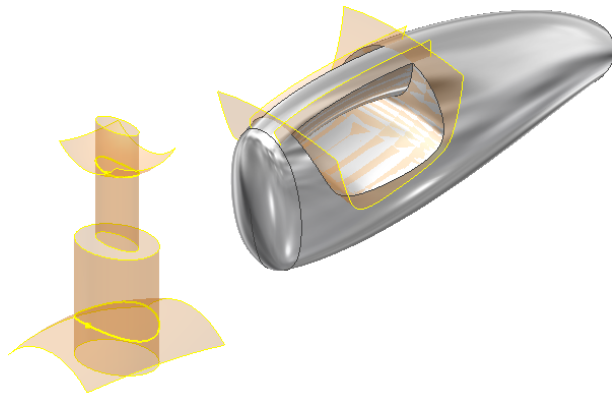
The following features require sketches, but do not create a base feature because they are dependent on existing geometry.

Rib

Creates a rib or web extrusion from a 2D sketch.
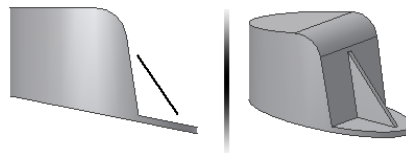
Use Rib to create thin-walled closed support shapes (ribs) and thin-walled open support shapes webs.

Emboss

Creates a raised (emboss) or recessed (engrave) feature from a sketch profile.
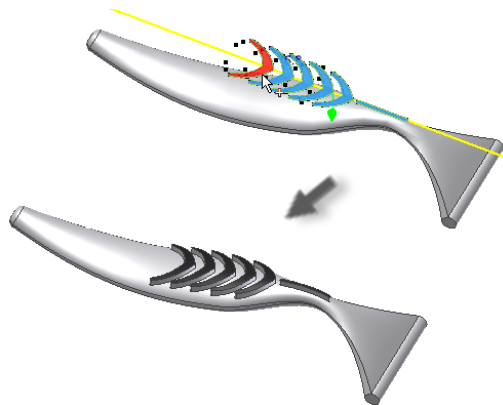
Decal

Applies an image file to a part face.

Use decal to add realism or to apply a label.
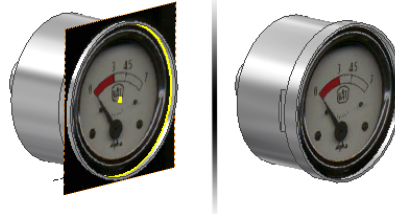
For more information	Location
Help topic	Search: "Plan and create sketches" "Sketch properties"
Tutorial	Parts 1 - Create Parts

Sketch Environment

When you create or edit a sketch, you work in the sketch environment. The sketch environment consists of a sketch and sketch commands. The commands control the sketch grid and draw lines, splines, circles, ellipses, arcs, rectangles, polygons, or points.

When you open a new part file, the sketch environment is active. The 2D Sketch button is selected, and the Sketch commands are available, along with a sketch plane on which to sketch. You can control the initial sketch setup by using template files, or settings in the Application Options dialog box, Sketch tab.

When you create a sketch, a sketch icon displays in the browser. When you create a feature from a sketch, a feature icon displays in the browser with the sketch icon nested under it. When you click a sketch icon in the browser, the sketch is highlighted in the graphics window.

After you create a model from a sketch, re-enter the sketch environment to change or start a new sketch for a new feature. In an existing part file, first activate the sketch in the browser. This action activates the commands in the

sketch environment. You can create geometry for part features. The changes you make to a sketch are reflected in the model.

For more information	Location
Help topic	Search: "Sketch Environment" "Application Options settings > Part tab" "Application Options settings > Sketch tab"
Tutorial	Work with Sketch Blocks

Sketch Blocks
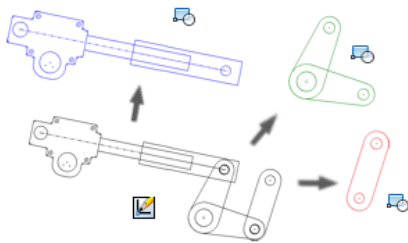
In many assembly designs, rigid shapes are repeated. You can use sketch blocks to capture such shapes as a fixed set, and place instances of the set where needed.

You can define nested sketch blocks and place flexible instances of these blocks. These flexible instances retain specified degrees of freedom that allow them to simulate kinematic subassemblies.

Sketch blocks are created in 2D part sketches and can be comprised only of sketch objects. Sketch block definitions are contained in the Blocks folder while sketch block instances reside under the parent sketch. You can control the appearance and format of block definitions and instances.

Use sketch blocks to represent components in your top-down design layout. After you create a sketch block, you can add instances of the block to your layout. This method for adding components in multiple locations in the design is quick and associative. Any changes to the block definition are propagated to all block instances.

For more information	Location
Help topic	Search: "Sketch blocks" "Top-down design"
Tutorial	Sketch Blocks

Sketch Constraints

Constraints limit changes and define the shape of a sketch. For example, if a line is horizontally constrained, dragging an endpoint changes the length of the line or moves it vertically. However, the drag does not affect its slope. You can place geometric constraints between:

■ Two objects in the same sketch.

■ A sketch and geometry projected from an existing feature or a different sketch.

As you sketch, constraints are automatically applied to the various sketch elements. For example, if the horizontal or vertical symbol displays when you create a line, then the associated constraint is applied. Depending on how accurately you sketch, one or more constraints could be required to stabilize the sketch shape or position. You can also add constraints manually to any sketch element.

Although you can use unconstrained sketches, fully constrained sketches result in more predictable updates.

For more information	Location
Help topic	Search: "Constrain Sketches"
Tutorial	Explore Sketch Constraints

2D AutoCAD Data in Sketches

When you open an AutoCAD® file in Autodesk Inventor, you can place 2D translated data:

■ On a sketch in a new or existing drawing.

■ As a title block in a new drawing.

■ As a sketched symbol in a new drawing.

■ On a sketch in a new or existing part.

You can import AutoCAD (DWG) drawings into a part sketch, drawing, or drawing sketch overlay. The entities from the XY plane of model space are placed on the sketch. In a drawing, certain entities, such as splines, cannot

be converted. You can choose to import AutoCAD blocks as Autodesk Inventor sketch blocks.

When you export Autodesk Inventor drawings to AutoCAD, the converter creates an editable AutoCAD drawing. All data is placed in paper space or model space in the DWG file. If the Autodesk Inventor drawing has multiple sheets, each is saved as a separate DWG file. The exported entities become AutoCAD entities, including dimensions.

You can open a *.dwg* file and then copy selected AutoCAD data to the clipboard and paste into a part, assembly, or drawing sketch. The data is imported at the cursor position.

For more information	Location
Help topic	Search: "3D sketch environment" "AutoCAD, using geometry in Inventor"

Placed Features

Placed features are common engineering features that do not require a sketch when you create them with Autodesk Inventor. You usually provide only the location and a few dimensions.

The standard placed features are shell, fillet, chamfer, face draft, hole, and thread.

The commands for placed features are located on the Sketch and Model tabs:

Fillet Places a fillet or round on selected edges loops, and features.

Chamfer Breaks sharp edges. Removes material from an outside edge and adds material to an inside edge.

Hole Places a specified hole in a part, optionally with thread.

Thread Creates regular and tapered external and internal threads on cylindrical or conical faces.

Shell Produces a hollow part with a wall thickness you define.

Rectangular Pattern Creates a rectangular pattern of features.

Circular Pattern Creates a circular pattern of features.

Mirror Feature Mirrors different type of features across a plane.

Dialog boxes define values for placed features, such as the Hole dialog box.

iFeatures

An iFeature is one or more features that you can save and reuse in other designs. You can create an iFeature from any sketched feature. Features dependent on the sketched feature are included in the iFeature. After you create an iFeature and store it in a catalog, you can drag it from Windows Explorer and drop it in the part file. You can also use the Insert iFeature command.

For more information	Location
Help topic	Search: "Placed features" "iFeature fundamentals"

Assembly Features

Assembly features are like part features, except that you create them in the assembly environment. They can affect multiple components in an assembly file, but the modifications do not alter the included component files. If assembly features are used, use LOD reps to exclude unnecessary components. The more participants, the bigger the file size and the longer it takes to calculate the feature. You usually suppress assembly features before saving.

Assembly features include chamfers, fillets, sweeps, revolved features, extrusions, holes, move face, rectangular feature pattern, circular feature pattern, and mirror. They also include the work features and sketches used to create them. The workflow and dialog boxes are the same as for part features. However, some operations are not available, such as creating a surface for extruded and revolved features.

You can edit, add to, suppress, or delete assembly features. You can also roll back the state of the assembly features and add or remove components that participate in the feature.

For more information	Location
Help topic	Search: "Assembly Features"
Tutorial	Assemble and Constrain Components

For more information	Location
Showme	Show me how to create an assembly feature

Work Features

Work features are abstract construction geometry that you can use to create and position new features when other geometry is insufficient. To fix position and shape, constrain features to work features.

Work features include work planes, work axes, and work points. The proper orientation and constraint conditions are inferred from the geometry you select and the order in which you select it.

The work feature commands provide on-screen prompts to help you with selection and placement. You can:

- Create and use work features in the part, assembly, sheet metal, and 3D sketch environments.

- Use and refer to work features in the drawing environment.

- Project work features into a 2D sketch.

- Create in-line work features to help you define a 3D sketch or position a part or assembly feature.

- Make work features adaptive.

- Turn the visibility of work features on or off.

- Drag to resize work planes and work axes.

For more information	Location
Help topics	Search: "Adaptive work features" "Work axes" "Work planes" "Work points"
Tutorial	Explore Sketch Constraints

Edit Features

In the browser, right-click a feature, and then use one of several options on the menu to modify the feature:

Show Dimensions	Displays the sketch dimensions so you can edit them.
	■ Change the dimensions of a feature sketch.
	■ Change, add, or delete constraints.
Edit Sketch	Activates the sketch so it is available for edit.
	■ Modify or create a new profile for the feature.
	After you modify a part sketch, exit the sketch and the part updates automatically.
Edit Feature	Opens the dialog box for that feature.
	■ Choose a different method to terminate the feature.
	■ Choose whether the feature joins, cuts, or intersects another feature.
3D Grips	Uses grip handles to drag a feature or face, or snap to other geometry to resize a feature. Arrows indicate the drag direction. The feature preview shows the expected results before you commit to the change.
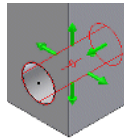
For more information	Location
Help topic	Search: "Features and feature termination"
Tutorial	Parts 2 - Create Base Parts

Assemblies

Assembly modeling combines the strategies of placing existing components in an assembly, and creating other components in place within the context

of the assembly. In a typical modeling process, some component designs are known and some standard components are used. Create the designs to meet specific objectives.

Place Components

In the assembly environment, you can add existing parts and subassemblies to create assemblies, or you can create parts and subassemblies in-place.

A component (a part or subassembly) can be an unconsumed sketch, a part, a surface, or any mixture of both.

When you create a component in-place, you can do one of the following:

■ Sketch on one of the assembly origin planes.

■ Click in empty space to set the sketch plane to the current camera plane.

■ Constrain a sketch to the face of an existing component.

When a component is active, the rest of the assembly is shaded in the browser and graphics window. Only one component can be active at a time.

Choose a fundamental part or subassembly, such as a frame or base plate, to be the first component in an assembly. Except for the first placed component, all placed components are unconstrained and ungrounded. You add the constraints you need.
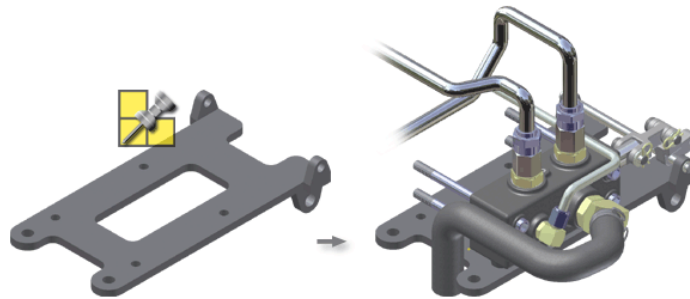
The first component you place in an assembly is automatically grounded (all degrees of freedom are removed). Its origin and coordinate axes are aligned with the origin and coordinate axes of the assembly. It is a good practice to place assembly components in the order in which they would be assembled in manufacturing.

When you create a component in the assembly context, the created component is nested under the active main assembly or subassembly in the browser. A sketch profile for the in-place component that uses projected loops from other components within the assembly, is associatively tied to the projecting components.
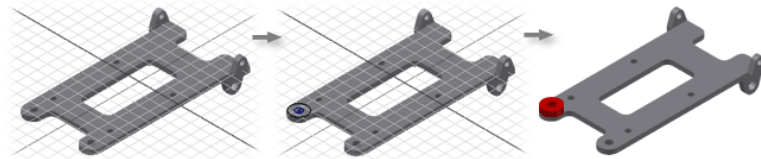
Drag Components into Assemblies

You can place multiple components in an assembly file in a single operation by dragging them into an open assembly window.

Drop the files over the graphics window where the assembly model is displayed. A single occurrence of each component is placed in the assembly file. The dropped components appear at the bottom of the browser in the receiving assembly.

For more information	Location
Help topic	Search: "Assembly components"
Tutorial	Assemble and Constrain Components

Assembly Constraints

Assembly constraints establish the orientation of the components in the assembly and simulate mechanical relationships between components. For example, you can:

■ Mate two planes.

■ Specify that cylindrical features on two parts remain concentric.

■ Constrain a spherical face on one component to remain tangent to a planar face on another component.

Each time you update the assembly, the assembly constraints are enforced.

Degrees of Freedom

Each unconstrained component in an assembly has six degrees of freedom (DOF). It can move along or rotate about each of the X, Y, and Z axes. The ability to move along X, Y, and Z axes is called translational freedom. The ability to rotate around the axes is called rotational freedom.

Whenever you apply a constraint to a component in an assembly, you remove one or more degrees of freedom. A component is fully constrained when all degrees of freedom (DOF) are removed. You are not required to constrain completely any component in an assembly in Autodesk Inventor.

To verify the DOF of components in an assembly:

■ Select Degrees of Freedom from the Visibility panel of the View tab.

■ Drag a component in the graphics window. Other components in the assembly will move based on existing constraints.

For more information	Location
Help topic	Search:
	"Assembly Constraints Overview"
	"Degrees of Freedom in Assemblies"
	"Plan Constraints"
Tutorial	Assemble and Constrain Components

Top-down Design
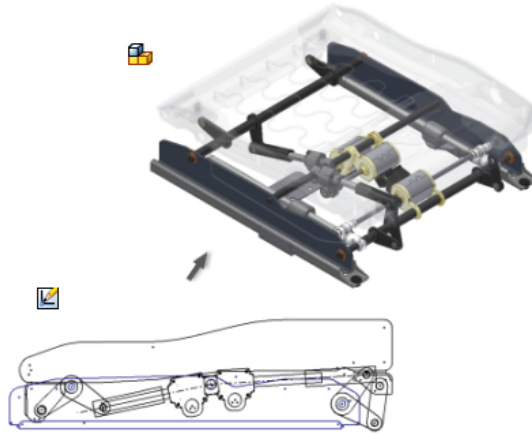
The top-down design technique (also known as skeletal modeling) centralizes control of your design. The technique enables you to update your design efficiently and with minimal disruption to your design documents.

Top-down design begins with the layout. The layout is a 2D part sketch that is the root document of your design. You create a layout that represents your assembly, subassembly, floor plan, or equivalent. In the layout, you use 2D sketch geometry and sketch blocks to represent the design components. You position these components, in the layout, to evaluate design feasibility.

Once you are satisfied with the state of your layout, you make components from the sketch blocks. This process, also known as push-derive, results in part and assembly files that are associated to the layout sketch blocks. When you change the sketch block definitions, your component files automatically reflect the changes.

Experiment with top-down design to experience the power of truly associative designs.

For more information	Location
Help topics	Search: "Top-down design"
Tutorial	Top-down Workflow

Create Subassemblies In-place

In the assembly environment, you can add existing parts and subassemblies to create assemblies or you can create new parts and subassemblies in-place.

A component (a part or subassembly) can be an unconsumed sketch, a part, a surface, or any mixture of both.

When you create a component in-place, you can do one of the following:

- Sketch on one of the assembly origin planes.

- Click in empty space to set the sketch plane to the current camera plane.

- Constrain a sketch to the face of an existing component.

When you create a subassembly in place, you define an empty group of components. The new subassembly automatically becomes the active assembly, and you can start to populate it with placed and in-place components. When you reactivate the parent assembly, the subassembly is treated as a single unit in the parent assembly.

Optionally, you can select components at the same assembly level in the browser, right-click, and then select Component ➤ Demote to place them into a new subassembly. You are asked to specify a new file name, template, location, and default bill of materials structure. You can then move components between assembly levels by dragging components in the browser.

Subassemblies can be nested many layers deep in a large assembly. By planning and building subassemblies, you can efficiently manage the construction of large assemblies. You can create subassemblies that match the intended manufacturing scheme to facilitate the creation of your assembly documentation.

For more information	Location
Help	Search: "Top-down, bottom-up, middle-out design"
Tutorial	Design Assemblies and Constraints
Skill Builders	Assemblies

Design Accelerator Components

Design Accelerator provides a set of generators and calculators to create mechanically correct components automatically from simple or detailed mechanical attributes you enter.

You insert components using Design Accelerator generators and calculators in the assembly environment. The generators and calculators are grouped according to functional areas. For example, all welds are together.

For more information

|	
Help topic	Search: "Design Accelerator"
Tutorials	Design Bolted Connections, Shafts, Spur Gears Connections, Bearings, V-belts Connections, Disc Cams, Compression Springs
Skill Builder	Design Accelerator

Design Mechanisms

A mechanism is defined as a design with one or more moving parts. Inventor provides numerous tools to assist you in creating and evaluating a mechanical design.

Use sketch blocks in a 2D part sketch to create a schematic layout of your mechanism. Create flexible, nested blocks and apply sketch constraints to define subassembly kinematics. Derive sketch blocks into component files and create other features to develop your 3D models. The components remain associated to their corresponding blocks and update to reflect any changes in block shape.

Use the following tools to evaluate a mechanism in the 3D environment:

■ Animate an assembly constraint and enable collision detection to determine the exact point of contact. For example, animate an angular constraint to evaluate the range of motion before contact occurs.
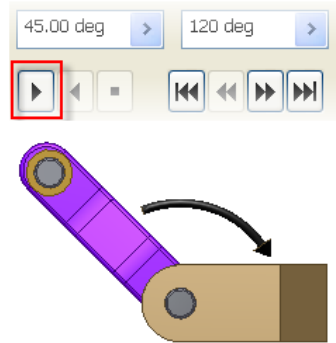
■ Create a Contact Set and add members as required to simulate physical contact between components and to determine the range of motion.
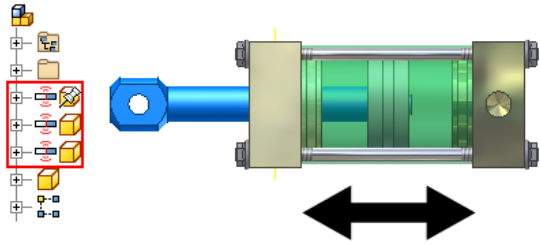
■ Use Positional representations to save a mechanism in various states such as maximum and minimum extension.

■ Use Inventor Studio to animate simultaneous or sequential movement.

- Use the Dynamic Simulation Environment to calculate displacements, velocities, accelerations, and reaction forces without the cost of a physical prototype.

- Use the Stress Analysis Environment to conduct structural static and modal stress analysis studies on the digital prototype.

For more information	Location
Help topic	Search: "Physical environment"
Tutorial	Animate Assemblies Explore Part Stress Analysis Explore Assembly Simulation

Check for Interference

In the physical product built from your design, two or more components cannot occupy the same space at the same time unless they are specifically designed to do so. To check for such errors, Autodesk Inventor can analyze assemblies for interference.

The Analyze Interference command checks for interference between sets of components or among the components in a single set. If interference exists, Autodesk Inventor displays it as a solid and displays a dialog box that contains the volume and centroid of each interference. You can then modify or move the components to eliminate the interference.
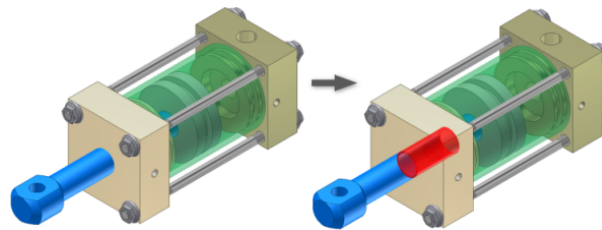
For more information	Location
Help topic	Search: "Check for interference between components"
Tutorial	Optimize Assemblies

iAssemblies

An iAssembly is a configuration of a model with a few or many variations called members. Each member has a set of unique identifiers, such as diameter or length. A member could have different components, such as a power train for a vehicle with several different engine sizes.

Create an iAssembly if you want to show different quantities for assembly components in a parts list. You can define the required parts list quantity for each iAssembly member.

You can manage iAssemblies from a table. In an iAssembly, you can replace one member with another member from the same factory by selecting a different row in the table. The bill of materials and parts list automatically update when you edit members.

For more information	Location
Help topic	Search: "iAssemblies"

Document and Publish Designs

3

During the process of creating digital prototypes in Inventor, there is often a need to communicate the design to individuals outside the design team. In Autodesk Inventor®, you can create the appropriate type of documentation for any consumer, such as customers or manufacturers. The document types available are:

- 2D drawings

- 3D CAD files

- Read-only files, such as DWF or PDF

- Photo realistic renderings

You can create the documentation at any stage during the process of creating digital prototypes.

Drawings

A drawing consists of one or more sheets that each contain one or more 2D drawing views and annotations. Drawings are associative to the digital prototypes. Any change to the model is automatically reflected in the drawing the next time you open it. You can create a drawing at any point in the design process, and it always reflects the current state of the digital prototype. Annotations can include dimensions, symbols, tables, and text.

Start Drawings

Drawings are created from a drawing template file. Autodesk Inventor includes standard templates (*.idw*, *.dwg*) stored in the *Autodesk\Inventor (version*

number)\Templates folder. The available templates are presented in the tabs of the New File dialog box.

Drawing templates can contain sheet formats, borders, title blocks, and sketched symbols. Templates also control the default styles and standards used for the appearance of views and annotations.

When you start a drawing, the title block, border, sheet size, and other elements come from the template.

Inventor Drawing Tips:

■ The template you select to create a drawing file determines the default sheet size, title block, border, and so on. You can change the sheet size, title block, and border after you create the drawing. The template controls the default styles and standards used for the appearance of views and annotations.

■ You can create customized templates and save them in the *Templates* folder. To set up a drawing template, open a template file from *Autodesk\Inventor (version number)\Templates*. Make your changes, and save the file with a new name in the *Templates* folder. The new template is available the next time the New File dialog box displays.

■ You can create different templates for the different sheets sizes you use, or create multiple sheet formats in a single template.

■ To customize a drawing sheet in a template, change the default sheet size and specify sheet orientation. Then modify the border and title blocks to fit the sheet. Available borders and title blocks are listed in the Drawing Resources folder in the browser.

For more information	Location
Help topics	Search:
	"Drawing Environment"
	"Templates for drawings"
	"Create drawings"
Tutorial	Prepare Final Drawings

Types of Drawing Files

Autodesk Inventor supports IDW and DWG file types for drawings. Both file types produce identical drawings. IDW files are the native Inventor format.

You can open them only in Inventor or Inventor View. This file type results in smaller file sizes.

The DWG file type is native to AutoCAD®. You can open DWG files in AutoCAD, Inventor, or DWG TrueView. If you create data using Inventor in a DWG file, you can modify the data only with Inventor. If you create data using AutoCAD in a DWG file, you can modify the data only with AutoCAD. If a downstream consumer of your Inventor data needs a DWG file, consider using DWG files as the default in Inventor.

For more information	Location
Help topics	Search: "Create drawing templates" "DWG Translation"

Create Views of Models

A drawing view is a 2D representation of a 3D digital prototype that is placed on a drawing sheet. The commands for views in Inventor are like view types in drafting. The two types of view commands in Inventor are author and modify. Author commands create new views. They are located in the Author panel of the ribbon. Modify commands change existing views. They are located on the Modify panel of the ribbon.

The first view placed on a drawing is a base view. Subsequent views are either children of the base view or additional base views. As you create views, they are listed in the browser with the sheet, title block, and border.
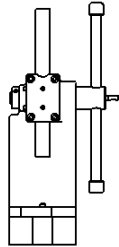
Types of Drawing Views
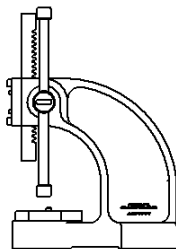
Base View
The first view created in a drawing. The base view is the source for subsequent views and controls the scale and alignment for them. You can create one or more base views on a drawing sheet.
You select the orientation of the view when you create it. The default orientations are based on the origin in the digital prototype.

Projected View

An orthographic or isometric view that is generated from a base view or other existing view. You can create multiple projected views in a single operation. The position of the cursor relative to the parent view determines the orientation of the projected view.

Projected views inherit the scale and display settings from the parent view. Orthographic projected views keep alignment with the parent view. The active drafting standard defines the first-angle or third-angle projection.
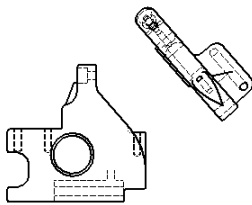
Auxiliary View

A view projected perpendicular to a user-selected line or edge. Use Auxiliary View to document features on inclined faces.

The position of the cursor relative to the parent view determines the orientation of the auxiliary view. Auxiliary views inherit the scale and display settings from the parent view.
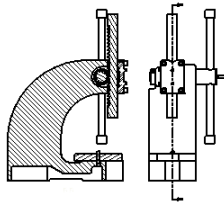
Section View

A view created by sketching a line that defines a plane used to cut through a part or assembly. You draw the cutting line when you create the view, or select it from a sketch associated to the parent view. The cutting line can be a single straight segment or multiple segments. The cutting line arrowheads on the base view automatically orient to reflect the position of the section view relative to the base view.

The crosshatching, section line, and labels are placed automatically.
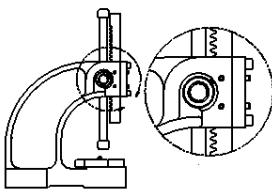
Detail View

An enlarged view of a specified portion of another drawing view. By default, the scale of the detail view is double the scale of the parent view, but you can specify any scale. A detail view is created without alignment to its parent view.

Autodesk Inventor labels the detail view and the area it is derived from on its parent view. You can set either a circular or rectangular fence for the detail.
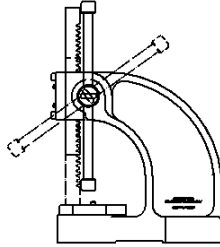
Overlay View

A single view that shows an assembly in multiple positions. Overlays are available for base, projected, and auxiliary views. The overlay view is created on top of the parent view.

Draft View

A view created from a 2D sketch in the drawing file. You can place a draft view and construct a drawing without an associated model. A draft view can provide detail that is missing in a model.

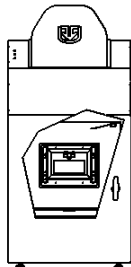# Drawing View Operations

Break

An operation that reduces the size of a model by removing or "breaking" irrelevant portions. Create a break in a view if the component view exceeds the length of the drawing, or contains large areas of nondescript geometry. An example is the center portion of a shaft.
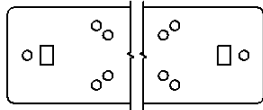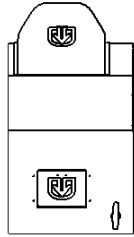
Dimensions that span the break reflect the true length.

Break Out

An operation that removes a defined area of material to expose obscured parts or features in an existing drawing view. The parent view must have an associated sketch that contains the profile defining the break out boundary.

Crop

An operation that provides control over the view boundary in an existing drawing view. The clipping boundary can be a rectangle or circle you create during the command, or a closed profile you select from a sketch.
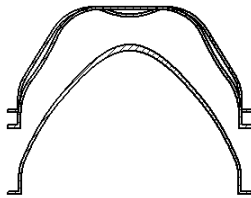
Slice

An operation that produces a zero-depth section from an existing drawing view. You perform the Slice operation in a selected target view. The slice lines are defined in a sketch associated to a different view.

Drawing View Tips

- You can edit views placed in drawings to change settings such as scale, hidden line display, thread display, and so on. If the edited view is a parent view, changes to the view parameters are reflected in dependent views.

- You can remove the association between a parent and a dependent view by editing the dependent view. Then you can set independent scale, style, and alignment for the dependent view.

- You can move a view by clicking and dragging the red border. You can move multiple views with a crossing selection window.

- Most dependent views are created with an alignment (vertical, horizontal, in position) to the parent. An aligned view can be moved only within its constraints. If the parent view is moved, the aligned view moves to maintain its alignment. You can manually break the alignment between the child and parent view.

- You can delete views that are no longer needed. If you delete a base view, dependent projected and auxiliary views can be deleted or retained. Section and detail views require a parent view and cannot be retained.

■ You can suppress views so that they do not display on the drawing sheet. Suppressed views are useful when one view is created only for creating a child view. The suppressed view can still be accessed in the browser.

| For more information	Location
Help topic	Search: "Drawing views"
Tutorial	Prepare Final Drawings
Skill Builders	Drawings

Exploded Views
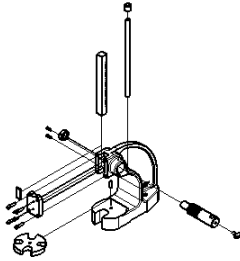
Exploded views are commonly used to describe assemblies by moving components out from their assembled position. Exploded views are often used to balloon an assembly using item numbers found in a parts list or bill of material. Exploded views are created by using a combination of assembly (*.iam)*, presentation (*.ipn*), and drawing (*.idw*, *.dwg*) files. A view of the assembly is created in the presentation file and the components are repositioned in the view. Drawing views are then generated from the presentation file.

For more information	Location
Help topic	Search: "Exploded views and presentations"
Tutorial	Create Exploded Views

Annotate Drawing Views

Drawing annotations provide additional information to drawing views to complete documentation of a digital prototype. The styles that correspond to the active drawing standard determine the appearance of drawing annotations. Drawing annotations are linked to model geometry and update accordingly with model changes. Once annotations are placed on the sheet, they can be moved using grips.

Types of Drawing Annotations
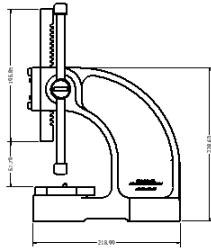
General Dimensions

You can create general dimensions in orthographic or isometric views. The geometry you select determines the dimension type and the options available in the right-click menu.

You can override the dimension text, which does not affect the model geometry.

You can change the dimension precision and tolerance, edit the leader and arrowheads, or modify the content of dimension text.
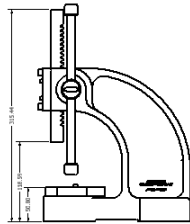
Baseline Dimensions and Baseline Dimension Sets

Creates multiple dimensions that display the orthogonal distance between the origin (base line) and selected edges or points. The first edge or point selected is the origin geometry. You can create individual dimensions or a dimension set.
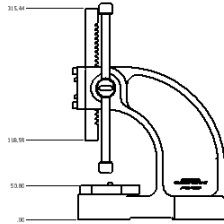
**Ordinate Dimensions and Ordinate Dimension Set**

Creates multiple ordinate dimensions in a single process. Ordinate dimensions automatically align as you place them. If dimension text overlaps, you can modify the dimension position or dimension style. You can create individual dimensions or a dimension set.
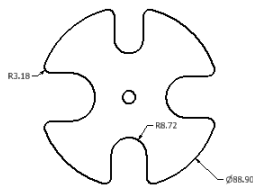
Retrieve Dimensions

Displays all model dimensions, or only dimensions related to selected parts or features. You select the dimensions to maintain in the drawing view.

Only model dimensions parallel to the view plane are available.

Model dimensions can be modified to manipulate the part file.
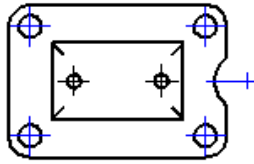
Center Marks

Center marks are added to the selected arc or circle. Center mark extension lines are automatically sized to fit the geometry. Center marks can be added individually or using the automated centerlines command.
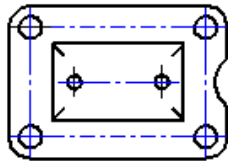
Centerlines

Creates centerlines for selected edges, at the midpoint for lines, or at the center point of arcs or circles. Creates a circular centerline when features form a circular pattern.

Autodesk Inventor supports three types of centerlines: bisector, centered pattern, and axial.
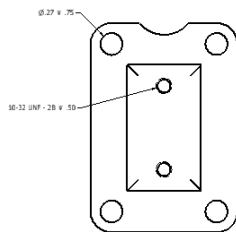
Hole/Thread Notes

Hole or thread notes display the information from hole, thread, and cylindrical cut extrusion features on a model. The style of the hole note varies depending on the type of feature selected.
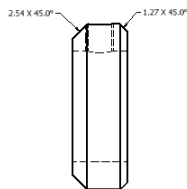
Chamfer Notes

Chamfer notes contain distance and angle measurements for selected model edges or sketched lines.

You can attach chamfer notes to angled edges in views and sketches. A chamfer edge and reference edge from different bodies, models, or sketches, must be part of the same view.
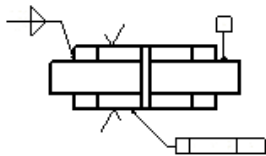
Symbols

Various types of symbols are available: Surface texture, welding, feature control frame, feature identifier, datum target, and datum identifier symbols. Symbols are created with or without a leader.

User-defined or sketched symbols are defined in the Drawing Re-sources and are placed like standard symbols. They are used to define custom symbols that are not available in Autodesk Inventor.

Bend Notes

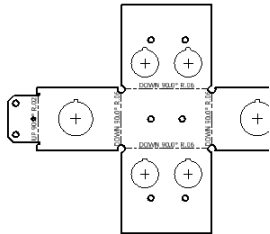A bend not adds fabricating information to sheet metal bend, contour roll, and cosmetic centerlines. Bend notes can be added to flat pattern views of sheet metal parts.

A bend note is associated with the selected bend centerline. The default placement of the bend note is above the selected bend centerline. It constrains the bend text to the midpoint of the centerline and offsets by the Origin Offset value from the Dimension Style.
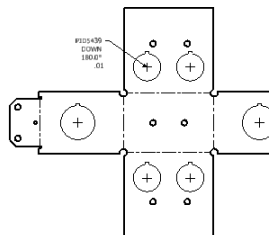
Punch Notes

A punch note includes data related to the punch feature: for example the punch ID, angle, direction, depth, quantity note, and so on. Punch notes can be added to flat pattern views of sheet metal parts.
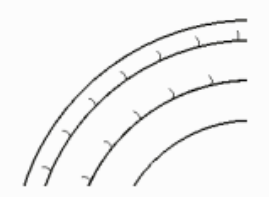
Caterpillars

Weld caterpillars are used to denote weld features in 2D views. You can add weld caterpillars manually using the Caterpillar command. Add them automatically from weld features using Get Model Annotations > Get Weld Annotations on the right-click menu.

End Fills

End fills are used to represent the filled region indicating the end of a weld bead. You can add them manually using the End Fill command or automatically from weldment models using Get Model Annotations > Get Weld Annotations.

Change end fill appearance through object properties.

Balloons

Balloons are annotation tags that identify items listed in a parts list. Balloons can be placed individually or automatically for all components in a drawing view. You can add balloons to a custom part after it is added to the parts list.
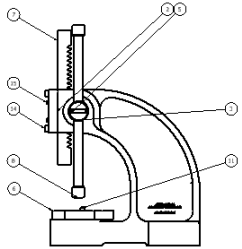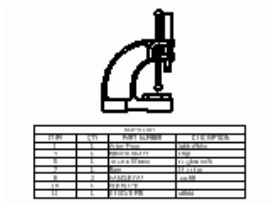
The balloon shape and value can be overridden using Edit Balloon on the right-click menu. You can combine balloons to use a single leader using the attach balloon options on the right-click menu.

Parts lists

Parts lists display data saved in the assembly bill of materials. The parts list can be modified to include different columns or overriden values. The bill of material data can be modified from the drawing file or the assembly file.

Custom parts can be added to the parts list to include items that are not modeled such as paint or grease.
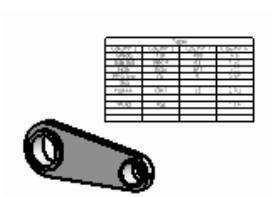
Tables

You can create a general, configuration, or bend table.

A general table can have a default number of rows and columns, or you can customize its size. The general table can reference external data from *.xls*, *.xlsx*, or *.csv* files, or you can enter any other type of data you need.

In drawings of iParts and iAssemblies, configuration table rows represent the members of the factory. You can specify the columns to include in the configuration table, such as exclusion status and values that are different among members.

A bend table is created if a sheet metal part is the source of the table. Bend tables contain bend information such as angle and radius.
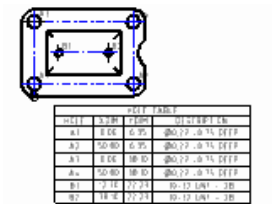
Hole Tables

Hole tables show the size and location of the hole features in a model. When a hole table is added, each individual hole receives a hole tag and a corresponding row is added to the table.

You can also add center marks, punch features, and cylindrical cut extrusion to a hole table by editing the table.

Revision Tables and Tags

Revision tables include information about design changes. Revision tables can be created for the entire drawing file or a single sheet. A revision tag marks an object changed by design revisions. The default revision level for the tag is the latest revision in the table. The revision level of the tag can be changed using the right-click menu.

Text or Leader Text

Use the Text command to add general notes to a drawing. General notes are not attached to any view, symbol, or other object in the drawing.

Use the Leader Text command to add notes to objects in a drawing. If you attach the leader line to geometry in a view, the note is moved or deleted when the view is moved or deleted.

For more information	Location
Help topic	Search "Drawing Annotations"
Tutorial	Prepare Final Drawings
Skill Builder	Drawings: Sketched Symbols: Stacking Parts List - Add a Custom Part

Styles and Standards

Inventor uses styles and standards to control the formats of views and annotation objects. The formats include dimension appearance, layers, linetypes, text styles, table formatting, and any other formatting related to annotation objects. Style and standard information is contained in a style library that is referenced by all documents. When you install Autodesk Inventor, you specify a default drafting standard which contains a set of styles. You can install and configure multiple standards and styles.

Styles and standards tips:

■ You can customize existing styles or create new styles. Copy an existing style, and make the desired changes to create a style. The style changes are saved in the current document and are not available to other documents until you save them to the style library.

■ You can store all or some of the style information in a drawing file or template instead of using the styles library. This method is useful when you make one-time overrides that you do not want to affect all drawings.

■ If a style is stored in a template, it is available only to future documents created with the template. Update manually the documents you created previously. With style libraries, a style definition is available in any document simply by refreshing the library.

■ Use the Object Defaults list to map drawing annotation objects to styles and drawing layers.

■ Link a customized style library to a project file (*.ipj*) to share styles among designers. All files included in the project then use the same styles for formatting. Documents are uniformly formatted, and updates are easy. When you update the main style definition in the library, all documents that use the style library can update their formatting.

For more information	Location
Help topic	Search: "Styles in drawings" "Configure the company standard styles using the Styles Editor"
Skill Builders	Drawings: Drawing Styles - Objects

Studio in Autodesk Inventor

Inventor Studio is a rendering and animation environment within Part and Assembly documents. You can produce both illustrative and realistic imagery of your part or assembly.

Inventor Studio images can be used to document products, present concepts to investors, clients, or your management team. The images you create can be used in a wide variety of places within the corporate environment. In addition, you can provide animations of how your design works. And, even better, with Video Producer (part of Studio) you are able to compose shots from multiple cameras and produce a more cinematic animation video.

As an environment within part and assembly documents, you simply choose when you want to enter the environment and do so. You can render an image using one of the standard lighting and scene styles or make new ones to suit your needs. You can save and share your custom lights, colors, and scene styles with others in your design community. Animations use the existing assembly constraints. All you do is specify a time period and what the intended end condition of the constraint is. A timeline provides easy access to adjusting the timing of each animated object.

Animating positional representations existing deep within the assembly requires extra time and effort. We recommend that you become familiar with Inventor Studio and the various representation options before you immerse yourself in this area.

Find out more within the Help content, books about Inventor, online resources of other Inventor users, and the Autodesk Newsgroup at *http://discussion.autodesk.com*.

For more information	Location
Help topic	Search: "Render and animate with Inventor Studio"
Tutorial	Render Models with Studio Animate Assemblies Create Positional Representations
Skill Builders	Studio: Animate Camera Advanced Camera Animation

Publish Designs

Digital prototype data can be published to various formats in Inventor. This data can include parts, assemblies, drawings, or a combination of these depending on the file type selected. To publish, use the Save Copy As command and select a file type, or use the various Export commands. Supported file types include:

- 2D and 3D DWF files

- CAD file formats including Parasolid®, Pro/ENGINEER®, and STEP

- 2D PDF files

- Image files including BMP, JPEG, PNG, or TIFF

DWF files are an Autodesk file type that can contain 3D data, 2D data, and bill of materials information. You can view DWF files in Autodesk® Design Review, a free download that contains tools to view, print, and mark up DWF files.

For more information	Location
Help topic	Search: "Publishing 3D data" "Publishing 2D data" "Publish to DWF" "DWF markup" "Export parts, assemblies, and more"
Web links	*autodesk.com/designreview* *dwfcommunity.autodesk.com* *autodesk.com/dwfwriter-download*

Manage Data

4

Autodesk Inventor® provides various means to share files within your internal workgroups, and with team members outside your organization. You can import and export files from and into other CAD software, and you can share Inventor files with team members who do not use CAD software.

Share Files in Work Groups Using Vault

Autodesk® Vault is a work group data management system for sharing design data across a project team. The vault provides a file management and version control system for all engineering and related data. It provides design team members with a central and secure collaborative environment.

Autodesk Vault in a shared environment consists of two components: the Vault server and vault clients. The server stores the master data files of all design information. The clients provide access to the files stored on the server.

Single Autodesk Vault Server
(Windows 2003 Server Preferred)
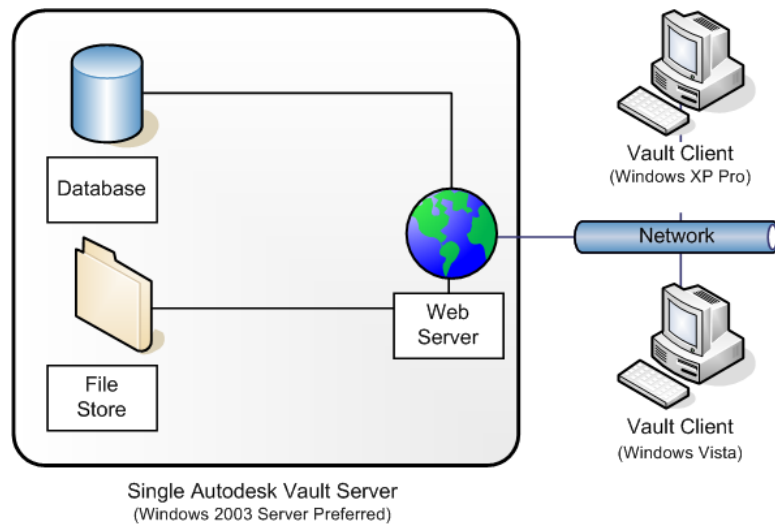
Design teams use Autodesk Vault for version control and to store and share all types of engineering files and related data. Files can be Autodesk Inventor, AutoCAD®, Autodesk® DWF™ (Design Web Format), FEA, CAM, or Microsoft® Office. They can be any other file used in the design process.

All versions of files that are checked into the vault are retained, along with any file dependencies, providing a living history of the project. Team members have access to the files and data stored on the server, and the history of files. They check out files to prevent more than one member from editing the same file at the same time. After a file is checked back into the vault, team members can refresh their local copies.

Autodesk Vault Add-ins for Design Applications

Add-in clients provide basic vault functions within the environment of a parent application, such as AutoCAD and Autodesk Inventor. Add-ins maintain application-specific data relationships when adding files to a vault.

There are add-ins available for Autodesk® and non-Autodesk design applications.

NOTE If an integrated client is available for a particular application, managing files using that client minimizes loss of data, like assembly relationships. We recommend that you use integrated clients whenever possible.

Microsoft Office Add-ins

The Microsoft Office Add-in performs basic vault functions on documents, spreadsheets, and other non-CAD data within any of these Microsoft Office applications: Word, Excel®, and PowerPoint®.

Copy Designs Using Vault

The Copy Design function in Autodesk Vault copies an Inventor design with all related files to create another design. Use Copy Design to copy an entire assembly structure, including all related 2D drawings and 3D models, to derive a new design. Copy Design preserves the relationships and can eliminate hours of rework. For example, an existing part and the related drawing file are copied and assigned new names. The new part and drawing are fully associative to each other. The dimensions and view layout from the original source document is preserved in the new drawing.

In the Copy Design dialog box, you can choose which parts of an existing design to copy, reuse, exclude, or replace. You can define a naming scheme for the files you copy to the new design. Alternatively, you can choose to add a prefix and a suffix to the file names automatically. If the original file names end in an integer, you can choose to automatically increment the names. You can rename presentation files and drawing files to match the names of their direct part or assembly source.

For more information	Location
Help topics	Search:
	"Learn about Autodesk Vault"
	"The Autodesk Vault Environment"
	"Tutorial: Performing Vault tasks in Autodesk Inventor"
	"Getting Started with the Autodesk Vault Add-in"
	"Autodesk Inventor and the Vault"

Share Files Externally

Team members outside your organization can access designs using Autodesk Productstream and Autodesk Design Review software. They can review and manage data throughout the design and manufacturing processes.

Autodesk Vault Manufacturing

Vault Manufacturing is a product data management (PDM) application that provides a modular and practical approach to controlling your design data. It bridges the gap between CAD data and the manufacturing process. You can:

- Track the life cycle of designs and materials used to manufacture a product.

- Manage what you make, buy, assemble, and deliver to customers.

Vault Manufacturing automates the process of tracking and managing the engineering release process. Items are managed through various working states such as work-in-progress, released, and obsolete. Bills of material (BOMs) coordinate the items in a total parts list. Change orders are submitted, reviewed, approved, and tracked through the routing process using Vault Manufacturing.

Vault Manufacturing Web Client is the Web browser-based application that gives access to the vault and its data to groups outside the engineering department. They can:

- Access all items and files in the Web Client, including released revisions.

- View, and print details of items and files in the Web Client.

For more information	Location
Vault Manufacturing Help	The separate Help in the Vault Manufacturing product.
Help topic	Search: "Installation and Deployment"

Autodesk Design Review

Using the free Autodesk Design Review software, team members who do not use CAD can access designs. They can review, mark up, measure, and track changes to designs and drawings. The markups and their statuses are saved in the DWF file.

For more information	Location
Autodesk Design Review software	Design Review Help.
Help topic	Search: "DWF markup"

For more information	Location
Web links	*autodesk.com/designreview*
	dwfcommunity.autodesk.com

Import and Export Data

To translate files, you open or import the files in Autodesk Inventor. You can also place part and assembly files as components in Autodesk Inventor assemblies and drag and drop part and assembly files into Autodesk Inventor.

In the open, import, and place components workflows, you can choose from specific import options to achieve the intended results. The import options are available when you select a file and click Options on the dialog box. Details of the import operation are contained in the translation report, under the corresponding third-party browser node.

AutoCAD Files

When you open an AutoCAD file in Autodesk Inventor, you can select the AutoCAD data to translate:

■ Model space, a single layout in paper space, or 3D solids.

■ One or more layers.

You can also place 2D translated data:

■ On a sketch in a new or existing drawing.

■ As a title block in a new drawing.

■ As a sketched symbol in a new drawing.

■ On a sketch in a new or existing part.

If you translate 3D solids, each solid becomes a part file containing an ASM solid body. Blocks are translated as sketched symbols.

You can import AutoCAD (DWG) drawings into a part sketch, a drawing, or a drawing sketch overlay. The converter takes the entities from the *XY* plane of model space and places them on the sketch. In drawings, you cannot convert certain entities, such as splines. You can choose to import AutoCAD blocks as Autodesk Inventor sketch blocks.

You can export Autodesk Inventor drawings to AutoCAD. The converter creates an editable AutoCAD drawing and places all data in paper space or model space in the DWG file. If the Autodesk Inventor drawing has multiple sheets, each is saved as a separate DWG file. The exported entities become AutoCAD entities, including dimensions.

You can open a DWG file and then copy selected AutoCAD data to the clipboard and paste into a part, assembly, or drawing sketch. The data is imported at the cursor position.

The options for importing and saving AutoCAD® files in Autodesk Inventor are:

■ Selection of layers.

■ Window selection of entities.

■ Saving files in DWG format.

■ Support for DXF files back to version 12.

■ Creation of AutoCAD®Mechanical files, if AutoCAD Mechanical is installed.

NOTE Mechanical Desktop files can be linked to Autodesk Inventor assemblies without importing.

For more information	Location
Help topics	Search: "DWG Translation" "Use your AutoCAD geometry" "Export drawing data to AutoCAD"

Import Files from Other CAD Systems
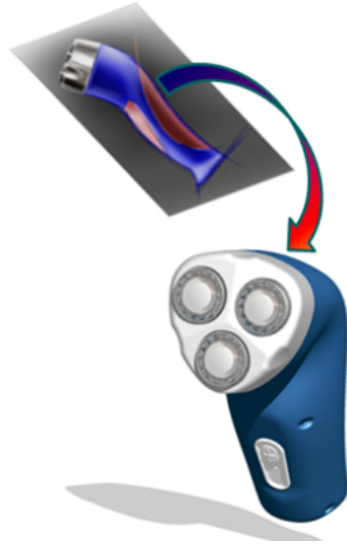
You can import part and assembly files from other CAD systems. The import operation does not maintain associativity with the original file, except when you associatively import Alias files. After you import the files, you can treat them as if they were originally created in Autodesk Inventor. You can import these files:

■ Alias

■ CATIA V5

■ JT

■ Pro/ENGINEER®

■ Parasolid®

■ SolidWorks™

■ UGS NX

You can also import SAT, STEP, IGES, and Mechanical Desktop (DWG) files.

For more information	Location
Help topics	Search: "Parts and assemblies from other CAD systems" "Import and use IGES data" "Import and use STEP data"

Export Files to Other CAD System Formats

You can export Autodesk Inventor parts, assemblies, and more to other CAD system formats. The export operation does not maintain associativity with the original Autodesk Inventor file. You can export these files:

- CATIA V5
- JT
- Pro/ENGINEER®
- Parasolid®

You can also export to SAT, STEP, IGES, DWF, and various graphic files formats.

For more information	Location
Help topics	Search: "Export parts, assemblies, and more" "Save, Save As, Export" "DWF markup"

Set Your Environment

5

The basics in this manual get you started using the Autodesk Inventor® software. References in For More Information tables throughout the manual guide you to Help topics, tutorials, and other resources that contain detailed information and specific instructions.

Find out more within books about Autodesk Inventor, online resources of other Autodesk Inventor users, and the Autodesk® Newsgroup at *http://discussion.autodesk.com*.

Along with the procedures in Help, the tutorials provide the step-by-step procedures that complement the information in this manual.

When you launch Autodesk Inventor, and before you open a file, the Get Started tab displays in the ribbon. The Get Started tab provides access to the many learning resources and customer involvement opportunities.

Commands and Tools

The Quick Access Toolbar (QAT) at the top of the screen is a set of common commands you can access across all environments. You can add commands to the QAT. Then, if necessary to accommodate numerous commands, you can dock the QAT under the ribbon.

The commands and tools for modeling are located on the ribbon below the QAT, organized by tab and by panels on each tab. The panel display on each tab is context dependent. In certain situations, a panel can be hidden. To check for hidden panels, right-click in the ribbon next to the panels. A list of the panels displays. Panels with no check marks are hidden. Select to add the check mark and display the hidden panel.

This illustration shows some of the panels on the Sketch tab.
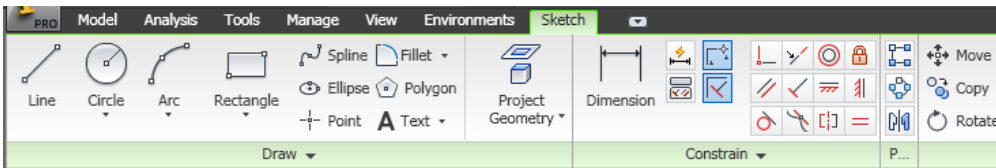
The arrows on some of the commands and panel name bars reveal more options.

The display of commands on the ribbon changes as you open and work in different types of files. Commands that are not accessible are shown as shaded, and you cannot select them.

Purpose or task drives the environments within Autodesk Inventor. The components of each environment are consistent in their placement and organization, including points of access for entry and exit. Unique colors identify tabs specific to a specialized environment so you can recognize the environment as you work.

To collapse the ribbon to the Panel name level, or to collapse it entirely, click the arrow to the right of the tab names.

When you start a file, the ribbon is docked at the top of the screen immediately below the QAT. View the video available in Help to see how to manipulate and customize the display of the ribbon.

For more information	Location
Help topics	Search: "Customize work environments in Autodesk Inventor" "View commands" "Custom command aliases" "Application Options settings"
Ribbon ➤ Get Started tab	Ribbon Introduction Ribbon Tutorial Command Locator Getting Started Guide Tutorials Show Me Animations

Environment Preferences

The options you select on the Application Options and Document Settings dialog boxes control the display of the environment. Access to the dialog boxes is on the Tools tab, Options panel.

Application Options

The settings in the Application Options dialog box control the look and feel of Autodesk Inventor. Various tabs control the color of your display, the behavior and settings of files, the default file locations, and various multiple-user functions.

Application options remain in effect until you change them.

Access to Application Options is available at the bottom of the Applications menu under Options.

Document Settings

The Document Settings dialog box controls the settings in individual files. Various tabs control settings for the active document. You can specify the active styles, units of measure, sketch and modeling preferences, bill of materials, and default tolerance.

For more informa- tion	Location
Help topics	Search: "Customize work environments in Autodesk Inventor" "Application Options settings"

Styles and Standards

The Style and Standard Editor provides options where you set your preferences for styles and standards. Access to the Style and Standard Editor is on the Manage tab, Styles and Standards panel.

You select a drafting standard when you install Autodesk Inventor. The default set of styles in standards control most objects used in documents, such as balloons, dimensions, text, tables, and so on. The default styles are usually

enough to get you started. Use the Style and Standard Editor to customize styles.

By default, actions such as creating or modifying styles affect only the current document. You can choose to save the style to the style library, a master library that contains definitions for all available styles associated with a drafting standard. Usually, a CAD administrator manages the style library. This practice ensures that no one replaces by accident the style definitions, used by all documents that use the drafting standard, with a custom style.

Style Libraries

Style libraries make it easy to share formatting conventions across projects because they contain the definitions of formatting objects. You can use a style library to update a style for all documents. For instance, you can revise the arrow heads of dimensions by editing the style and saving the revision to the master style library. All documents that use that drafting standard have access to the library and any new or changed styles that are added to it.

For more information	Location
Help topics	Search "Work with styles" "Styles in drawings"
Tutorial - Use Sheet Metal Styles	Tutorials link from Help home page.

Views of Models

The View tab contains commands for viewing models.
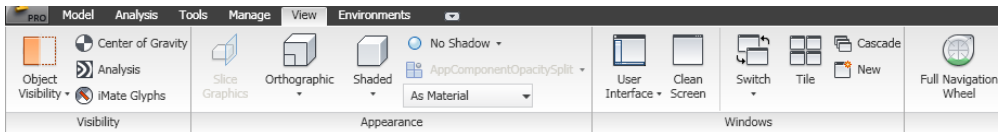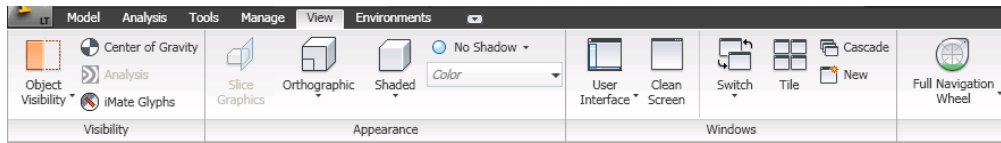
When ViewCube and NavBar are selected, they display in the upper right corner of the graphics window.

The NavBar contains the basic view commands.

For more information	Location
Help topics	Search: "Overview of the ViewCube" "Navigation Tools" "Views of models"

Templates

Once you activate Autodesk Inventor, you can open an existing file or start a new file. Templates are available on the Application menu under New. You can choose from several templates with predefined units. Use the tabs to select your standard.
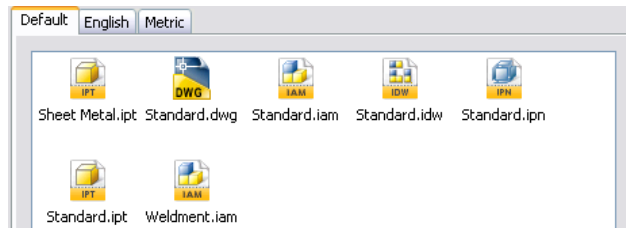
Templates are stored in the following directories in the *English* or *Metric* subdirectories.

■ Windows® XP: *Autodesk\Inventor(version number)\Templates*

■ Windows Vista®: *C:\Users\Public\Documents\Autodesk\Inventor(version number)\Templates*

Subdirectories in the *Templates* directory are displayed as tabs in the Open New File dialog box. You can create and save custom templates in the *Templates* directory..

For more information	Location
Help topics	Search: "Getting started" "Create files from templates"

Projects

A project represents a logical grouping of a complete design project. A project organizes your data by maintaining information about where design data is stored, where you can edit files, and maintains valid links between them. Projects are essential when you work in a team, work on multiple design projects, and share libraries among several design projects.

Autodesk Inventor supports two types of projects:

■ Single-user Project

■ Vault Projects (if Vault is installed)

A Project Editor is provided for you to create and edit project. In the Project Editor, you specify the project type, the default workspace, and library names and locations. Set the remaining project options to suit your design environment, and then specify project search paths.

Access to the Project Editor is provided:

■ On the Applications menu, under Manage.

■ In the Open dialog box.

■ On the Windows Start button, click Programs ➤ Autodesk ➤ Autodesk Inventor ➤ Tools.
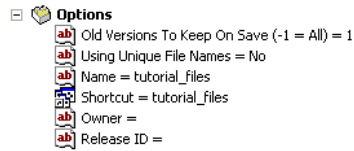
Vault Projects

We recommend that you use Vault projects to collaborate on projects with multiple designers. Common files are stored in a vault and are never accessed directly. Each designer has a personal project that defines where the files are copied for viewing and editing. The vault also maintains version history of files as well as additional attributes.
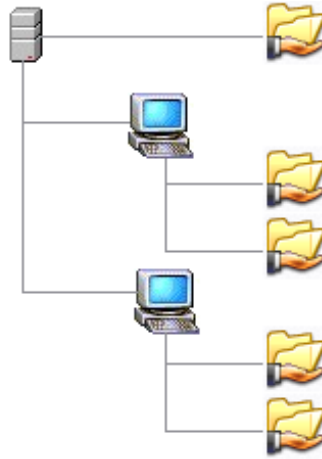
To use the vault project, Autodesk Vault software must be installed. A different dialog box opens so that you can create a Vault project. Characteristics of a vault project include:

■ Designers never view or work directly on the vaulted version of a file.

■ Each designer uses a project file that defines a personal workspace where Autodesk Vault copies the vaulted files for viewing and editing.

■ Changes to files made by other designers, and checked back into the vault, are not visible until you refresh your files to get the latest version in your workspace.

- Autodesk Vault maintains copies of all of the previous checked-in versions of data files. It stores additions about edit history, file properties, and file dependencies in its database.

- You can set up queries on file properties, track file references, and retrieve past configurations.

For a vault project, create a workspace at a path relative to the project file folder (such as .\ or .\workspace), and no other editable locations.

Default Projects

When you install Autodesk Inventor, it creates a "Default" project, a "samples" project, and a "tutorial_files" project automatically. If you do not create a project or specify a different project, when you start working in Inventor a default project is automatically active. Your files are saved to the Default project.

The default project does not define an editable location. However, you can use it to create designs immediately and save files anywhere without regard to projects and file management. Generally, you use the default project for experimentation only, not actual design work. It is easier to set up a project before you start designing. It is more difficult to migrate your files to a project when the design gets complex.

New Projects

Set up your file structure and understand who can access the file data before you create a project.

Set the project type when you create or edit a project. The type determines where files can be edited and saved, who has access to files, and check in and check out behavior.

The Project Wizard creates a workspace in the same folder as the project file. If you change that setting, keep the workspace as subfolders of the folder containing the project file.

This table summarizes recommendations for each project type.

Project type	Single User	Vault
Included file	None	None

Workspace locations	One defined at .\	One defined at .\
Workgroup locations	None	None
Libraries	One or more	One or more *not* nested under workspace

The default projects folder location is *My Documents/Inventor*, but you can change it to a different location.

For more information	Location
Help topic	Search: "Learn about projects"
Tutorial	Work with Projects

Learning Resources

Many opportunities for learning are available on the Get Started tab of the ribbon, and on the Help home page in the Autodesk Inventor software application.

New Features Workshop

New Features Workshop is a resource for all users. It is listed on the home page of the Help, and on the Get Started tab of the ribbon in Autodesk Inventor. It contains a description and illustration of each new feature in that particular version of Autodesk Inventor software.

Integrated Help

The Help is a comprehensive and convenient resource for learning about the Autodesk Inventor software.

In the Help browser, use the tabs to find topics using your favorite method.

On the home page, specify the type of user you are, and link to Help topics, additional information, resources, and services.

Many Help topics include example graphics and animations of functionality.
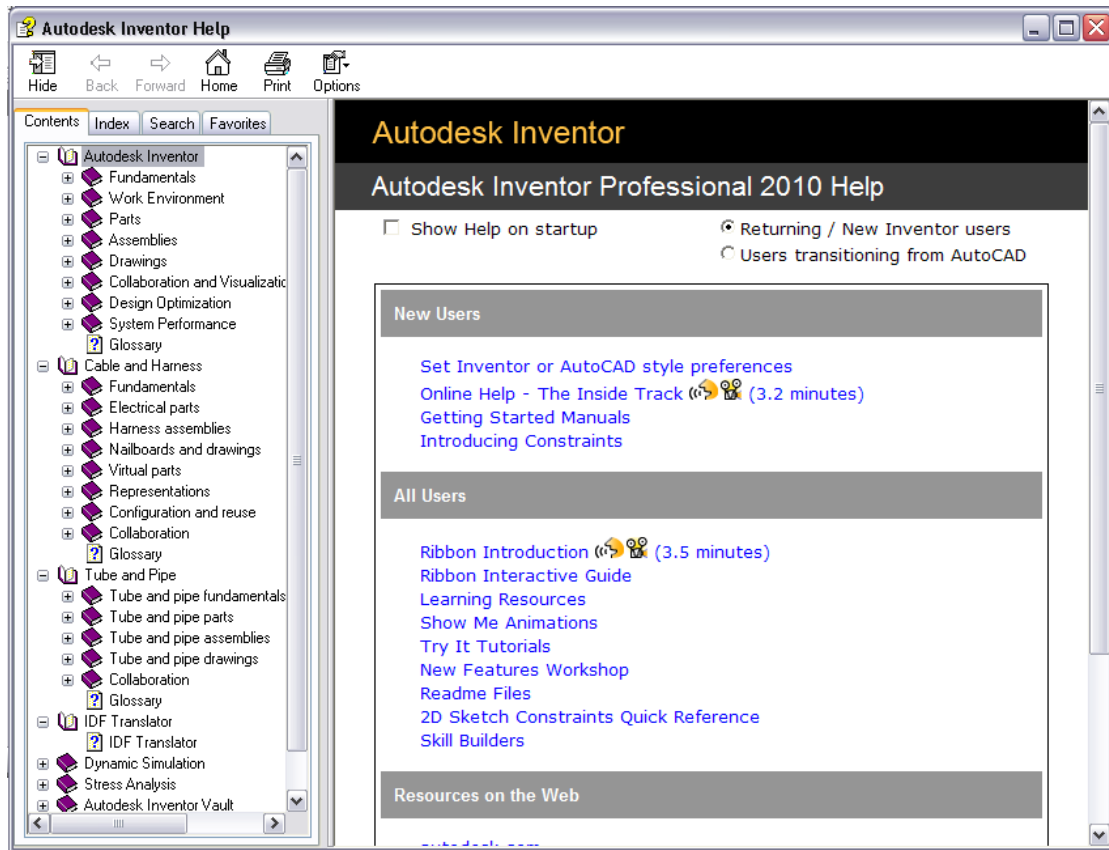
Tutorials

The tutorials you link to from the Help home page and the Get Started tab are a comprehensive set of hands-on lessons. The tutorial set is organized into three categories: fundamental, general interest, and specific interest. You can learn to be productive quickly, whether you are new to Autodesk Inventor or transitioning from AutoCAD.

Skill Builders

Skill Builders help you sharpen your skills in various specific areas of functionality. Use the link on the Help home page to navigate to the Skill Builders Web page.

For more information	Location
New Features Workshop	Link from Help home page
Help topics	Search: "Find the Information you need" "Autodesk Inventor Learning Resources"
Online Help: The Inside Track	Help home page
Skill Builders Web page (internet connection required)	Link from Help home page Ribbon Get Started tab
Tutorials	Link from Help home page Ribbon Get Started tab

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