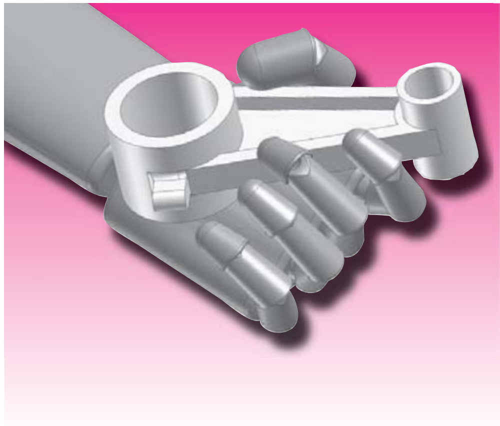


Chapter 1

Introduction to Autodesk Inventor

Objectives

After completing this chapter, you will be able to:

- Define a feature in Inventor.
- List the types of features in Inventor.
- Explain how to edit a part.
- Define an assembly in Inventor.
- Explain how to model motion in Inventor.
- Explain the engineer's notebook.

A Journey Begins

Inventor is a mature parametric solid modeling program built for mechanical design and engineering. It is in wide use throughout the world. With this program, you can design 3D models of complex parts, find their engineering properties, and create dimensioned detail drawings. Then, you can combine the parts into assemblies along with standard components, such as fasteners, from a built-in library. Assemblies can be animated to study their motion and to check for part interference. Assembly drawings with parts lists and balloons are easy to create and, like part drawings, are directly related to the model. In Inventor Studio, you can create realistic renderings and video files of animated assemblies.

This book uses a process-based approach. This offers a measured pace for the learner. A wealth of examples and exercises, along with the intuitive nature of Inventor, will allow you to become proficient and confident using Inventor with a moderate amount of study and practice.

Feature-Based Modeling

To explore the *feature-based* aspect of Inventor, refer to **Figure 1-1.** This is a part tree in the **Browser Bar**, or **Browser**, for a completed part. The **Browser** is an important part of the user interface. The ***part tree*** in the **Browser** lists the pieces and processes that make up the part. Inventor refers to these pieces and processes as ***features.*** If you think about any "single" part, it is really the end result of several features. The .ipt file extension is used for Inventor part files.

The part tree in **Figure 1-1** shows several features that together form the actual solid geometry of the final part. These features are named and have a colored icon next to their name that represents the type of feature. In this example, features include Hole for Outlet, Cleanup, Bolt Hole, and Bolt Pattern. These descriptive names were entered by the designer. By default, a feature is created with a generic name that represents the feature, such as Extrusion1 for an extrusion or Revolution2 for a revolution.

Work features are used for construction purposes. When created, they have names with the word Work in them, such as Work Plane2 and Work Plane3. The designer may elect to give work features descriptive names, such as Work Plane for Extrusion. Work features are not typically displayed in the graphics window, except when needed. Features that are not visible in the graphics window are grayed out in the **Browser**. The feature can be made visible as needed.

All of the features are listed in the part tree in the order in which they were created, with the first feature at the top. Features may be reordered by simply picking and dragging them to a new position in the part tree. Reordering features may alter the part. In addition, features cannot be moved above other features on which they are based. For example, if a hole has a fillet applied to it, the fillet feature cannot be moved above the hole feature. The last feature to be listed in any Inventor part tree is the End of Part marker. Its icon is a red octagon with a white X. This serves as an "end-of-file" marker for the part.

Features can be edited on an individual basis to change their size, shape, and, in some cases, location on the part. Features can also be suppressed. This means that the features are still in the part tree, but their effects on the part are not applied. To permanently remove a feature from the part, it is simply deleted. Features can even be exported for use in other parts.

The part that you are creating is never really edited; its *features* are edited. Since the part is made up of features, as the features are edited, the part is altered. Inventor is truly a feature-based solid modeling program and features form the heart of the system.

Inventor's features can be broken down into three categories—sketched, placed, and work. These categories are introduced in the next sections. As you work through this book, you will become very familiar with features and the tools used to create and edit them.

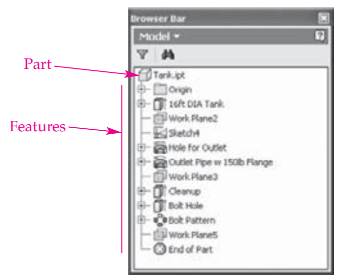
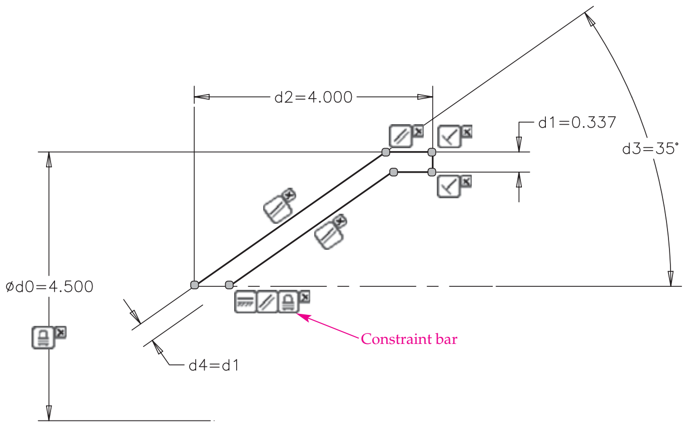


Figure 1-1. The part tree is displayed in the **Browser Bar**, or **Browser**. The tree contains all of the features that make up the part.

Figure 1-2.
This is a fully dimensioned and constrained 2D sketch.



Sketched Features

A 2D ***sketch*** forms the basis for a sketched feature. This sketch can be drawn, or sketched, as any 2D geometry from lines, arcs, circles, and so on. The sketch is then dimensioned to exact size. Furthermore, the geometric relationship of the sketch geometry is constrained to a final shape. See **Figure 1-2.** A ***fully constrained sketch*** completely describes the size, shape, and location of the sketched geometry so Inventor cannot inadvertently change the design in future operations. You will learn much more about sketches and constraints in later chapters.

Placed Features

Placed features are not based on a sketch. Instead, the designer uses tools to "place" features onto an existing part, which is a collection of features. Placed features are similar to machining processes and can be thought of as operations performed on an existing part. Examples of placed features include fillets, chamfers, and holes.

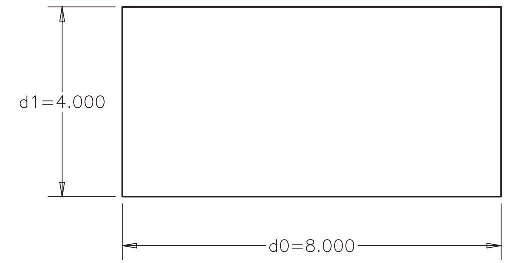
Work Features

Work features are used for construction purposes. Work features include work planes, work axes, and work points. In a 3D environment, sometimes there is nothing on which to base a 2D sketch except a work plane. The work plane serves as the "drafting table" for the sketch's "paper." A work axis can serve as a centerline of revolution. A work point can serve as an anchor point for a 3D path.

Parametric Modeling

As you have seen, an Inventor part is made up of features. This is why Inventor is considered a *feature-based* solid modeling program. However, Inventor is also a *parametric* solid modeling program. A ***parametric model*** contains parameters, or

Figure 1-3.
A rectangle is sketched and dimensioned.



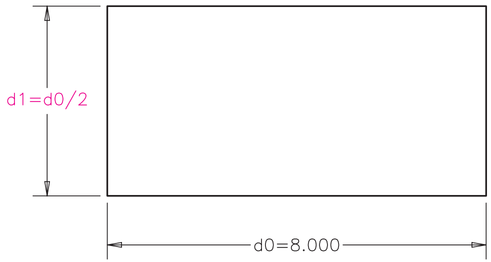
dimensions, that define the model. In Inventor, the features that make up a part have dimensions that control the size, shape, and location of the features. By altering the dimensions (parameters), the features are altered.

For example, suppose you need to design a rectangular plate. The preliminary design indicates the plate is 8″ × 4″ × .25″. To create the plate, you first sketch a rectangle that is roughly 8″ × 4″ on a sketch plane (the "paper"). Then, you apply a horizontal and a vertical dimension to the sketch and edit the values to 8″ and 4″. See **Figure 1-3.** The sketch completely describes the shape and size of the plate's top view. Next, you extrude the sketch .25″ to fully describe the part in three dimensions. See **Figure 1-4.** Since Inventor is a parametric modeler, you can now alter any or all of the three dimensions (parameters) to change the part.

Another powerful aspect of Inventor's parametric modeling is the ability to establish relationships between dimensions and features. For example, suppose the plate described above will always be half as wide as it is long, no matter what the length dimension is. An equation can be entered as a dimension to accomplish this. Every dimension, or parameter, in Inventor has a unique name. By default, the first dimension name is `d0`, the second is `d1`, and so on. As you will learn, these can be renamed to descriptive names as needed. Now, since the vertical distance (`d1`) needs to be one-half of the horizontal distance (`d0`), instead of entering a number for the dimension, the simple equation `d0/2` is entered for the value of `d1`. See **Figure 1-5.** As `d0` (the horizontal distance) is altered, `d1` changes so the vertical distance is one-half of the horizontal distance. This is known as *specifying design intent.* It is the intention of the designer that this plate always maintain its aspect ratio of 2:1 length to width. The sketch is thus dimensioned to reflect this design intent.

Figure 1-4.
The sketch from Figure 1-3 is extruded .25″ to create a solid part.

Figure 1-5.
An equation that is based on the length dimension is entered for the width dimension.



As mentioned, the default dimension names (d0, d1, etc.) can be renamed. Descriptive names, such as Length and Width, are meaningful to the drafter and designer. A year after the part is created, anybody can open the part file and instantly know what the dimension controls. Also, designers and drafters are almost always part of a team. Using descriptive names allows other team members to interpret your design intent. This is further enhanced by adopting a standard naming convention in your department or company.

Another use of named dimensions is to "manufacture" different versions of a part based on data in a spreadsheet. See **Figure 1-6.** As the part is "manufactured," the designer is prompted to enter values for various parameters. The prompts are based on the dimension names. Therefore, Pipe Length is a meaningful name, where d0 is not.

Assembly Modeling

Inventor does a great job modeling parametric parts. However, one of the most powerful aspects of Inventor is the ability to create assemblies. As a part is a collection of features, an assembly is a collection of parts. See **Figure 1-7.** Each part is created and saved. Then, the parts are *placed* into an assembly file. Parts are only referenced in the assembly. The part files remain separate. Changes made to the part file are reflected in the assembly. Finally, the spatial relationships between parts are defined, or constrained, within the assembly file. An assembly file has an .iam file extension.

A partially-constrained part can be dynamically dragged to analyze its movement within the assembly. The extents of its movement is the part's ***work envelope*** in the assembly. You can also move parts in an assembly and then measure distances to determine design data for additional parts.

Figure 1-6.
By using descriptive names, you can build a spreadsheet that is used to "manufacture" different versions of the part.

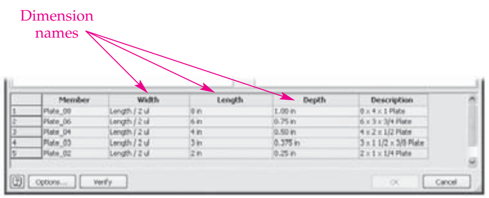
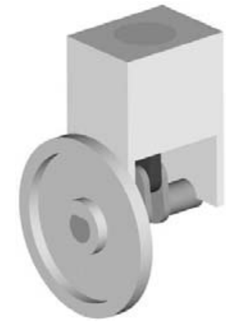


Figure 1-7.
This is a complex assembly. A number of individual parts were placed into the assembly and constrained to finish the assembly.



Modeling Motion

Assembly constraints can be "driven," or animated. The numeric values used in the assembly constraint can be dynamically changed over a specified range to model a part's movement within the assembly. This allows you to animate the motion of an assembly. Several types of motion can be animated:

- Rotational (gears).
- Rotational-translational (rack and pinion).
- Translational (cam and cam follower).

2D Drawings

Prints of 2D drawings are always required for the machinists, assemblers, and other workers in the shop. These drawings must follow accepted drafting conventions for lineweight, linetype, and symbol use. Inventor provides tools for creating 2D drawings of parts and assemblies. Inventor drawings have an .idw file extension. Parts and assemblies are referenced into the drawing and displayed using orthographic projection rules. Changes made to the part are automatically reflected in the drawing.

Presentations and Animations

If a picture is worth a thousand words, then how many words is a movie worth? A 2D drawing will always be required. However, you can capture to digital video the process for building an assembly. This is called a ***presentation***. See **Figure 1-8.** Inventor uses an .ipn file extension for presentation files.

The assembly file is referenced into the presentation and always reflects the current state of the assembly. The animated presentation can be distributed to others or saved as an AVI file to be viewed on workstations not equipped with Inventor. Presentations provide a very effective form of communication.

Figure 1-8.
An animated assembly presentation, such as this one, can be saved as an AVI file and played in Windows Media Player.

Another feature of Inventor, called Inventor Studio, allows you to create high-quality renderings and animations. You can add materials and lighting to the part or assembly model. This allows you to create photorealistic renderings. In addition, you can animate parts in an assembly and save the animation to a digital video file.

Engineer's Notebook

There is a handy little utility in Inventor that allows you to annotate a part in a notebook-style environment. This allows you to add to a part or feature notes that are not intended to be part of the final 2D drawing. See **Figure 1-9.** The engineer's notebook is meant to act as a communication tool between those collaborating on the design. Multiple notations may be made on a single part or assembly. The notes appear in the **Browser** in a separate branch in the part or assembly tree. An icon also appears in the graphics window near the annotated part. The engineer's notebook is discussed in detail in Chapter 2.

Design Assistant

In the process of creating parts, subassemblies, and an assembly, you may end up with hundreds of files. **Design Assistant** is a utility that acts as a document management system, **Figure 1-10.** It can track items such as revision number, status of design, and iProperties. When accessed through Windows Explorer, **Design Assistant** is a great utility for managing Inventor files.

Another very useful utility, called **Pack and Go**, allows you to select an assembly and it will find all of the parts, subassemblies, design views, etc., in the assembly and copy them to a destination folder. Then, you can zip all of the files and e-mail the zipped file.

Figure 1-9.
The engineer's notebook can be used to share comments that are not actual drawing annotations.

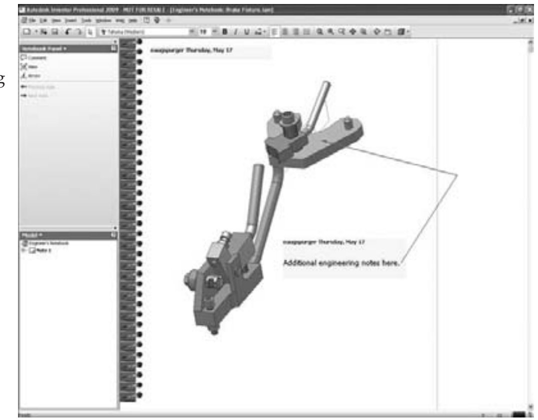
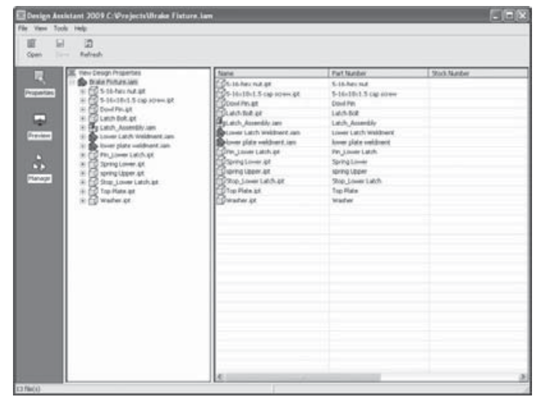


Figure 1-10.
Design Assistant serves as a document management system.



Chapter Test

Answer the following questions on a separate sheet of paper or complete the electronic chapter test on the Student CD.

1. What are *features* in Inventor?
2. How is Inventor a feature-based modeling program?
3. What are the three types of features in Inventor?
4. Which type of feature is used for construction purposes?
5. What is the basic process for editing a part?
6. How is Inventor a parametric modeling program?
7. Give one example of where you would establish a relationship between a part's parameters.
8. What is an *assembly?*
9. How can motion be modeled in Inventor?
10. What is the purpose of the engineer's notebook?