

Autodesk Inventor Tutorial 1

Introduction to Autodesk Inventor

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AUTODESK INVENTOR FUNCTIONS

A product usually consists of a number of component parts. Using computer as a tool to design and manufacture products, you construct computer models to represent each individual component of the product and put the computer models together to form a virtual assembly to explore and evaluate the integrity of the design. To illustrate how the components of an assembly are related to each other, you explode them apart. Although it is very common to use electronic data of computer models directly in downstream computerized manufacturing operations, there are times when conventional 2D engineering drawings are required. To meet this requirement, you output engineering drawings from the computer models of the individual parts, assemblies, and exploded assemblies. Serving these design requirements, Autodesk Inventor has four basic functions: constructing 3D parametric solid parts, assemblies of solid parts, exploded presentations of assemblies, and engineering drawings.

Constructing Solid Parts and Sheet Metal Parts

The prime function of Autodesk Inventor is to construct computer models in the form of 3D parametric solid parts and 3D sheet metal parts that represent a component part in the computer. Figure 1–1 shows a solid part for the model of a component of a radio controlled toy car.

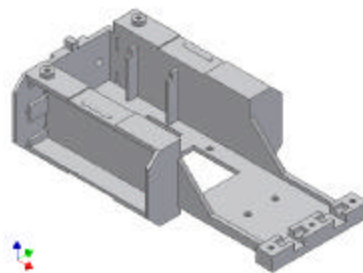


Figure 1–1
Solid part of the chassis of a scale model car

Sheet metal parts are a special kind of solid part. You make a sheet metal component by cutting and folding a sheet of metal of uniform thickness. To meet the manufacturing requirement of providing rounded bends at the joints of faces, relieves at the bends, hems at the edges, and seams at the joints, you need a special kind of solid modeling tool. Figure 1–2 shows the model of a sheet component.

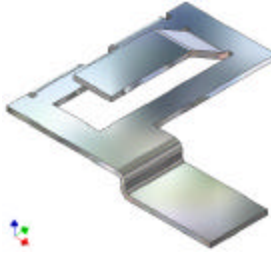


Figure 1–2
Sheet metal component

Constructing Assemblies of Solid Parts

The next function of Autodesk Inventor is to construct a virtual assembly of 3D solid parts. An assembly is a device consisting of a number of component parts. Figure 1–3 shows the assembly of the axle of a toy car.



Figure 1–3
Assembly of the axle of a toy car

Constructing Exploded Presentation of Assemblies

The third function of Autodesk Inventor is to construct an exploded presentation of assemblies. To illustrate how various parts of an assembly are put together, you construct a presentation of an assembly. In the presentation, you explode or tweak the components apart. Figure 1–4 shows that exploded presentation of the assembly shown in Figure 1–3.

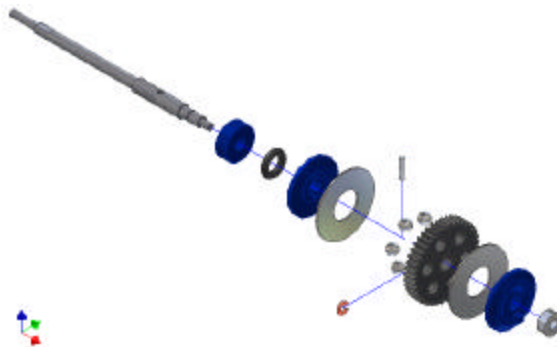


Figure 1–4
Presentation of the assembly of the rear axle

Constructing Engineering Drawings

The fourth function of Autodesk Inventor is to construct engineering drawings. Engineering drawing is an engineering communication tool that depicts a 3D design in 2D engineering drawing views. You specify a solid part or an assembly and the computer application can automatically generate 2D orthographic views of 3D solid parts, sheet metal parts, and assemblies, flat patterns of sheet metal, and exploded views of assemblies. Figure 1–5 shows the engineering drawing derived from the solid part of a toy car body.

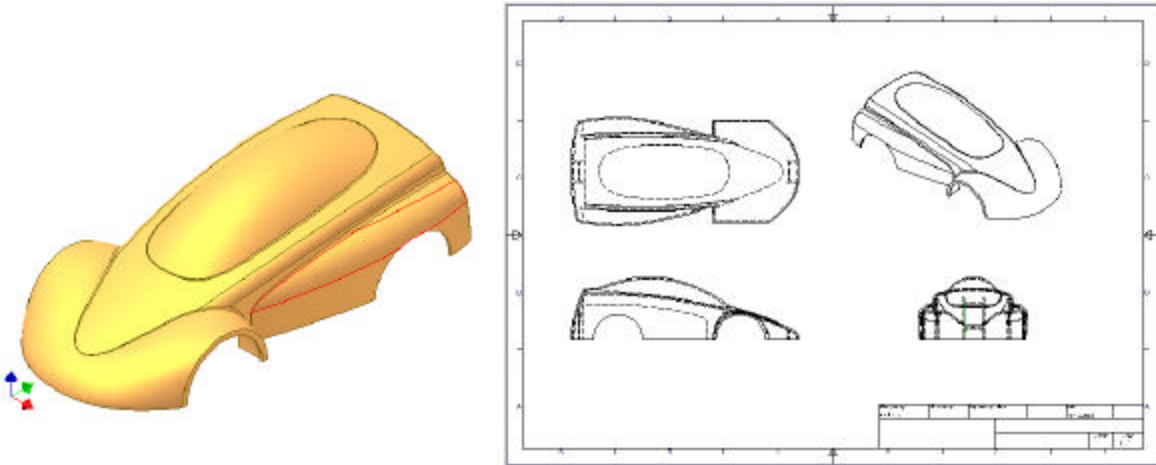


Figure 1–5
Toy car body and its engineering drawing

AUTODESK INVENTOR FILE TYPES

To cope with the four design functions, Autodesk Inventor uses four kinds of files:

Part files	for constructing solid parts
Assembly files	for assemblies of solid parts or sub-assemblies
Presentation files	for exploded views of an assembly
Drawing files	for engineering drawings

Part Files

You construct a 3D solid part or a sheet metal part in a part file; the file extension is .ipt. A part file stores the definition of the parametric 3D solid part.

Assembly Files

To construct an assembly or a sub-assembly, you use an assembly file; the file extension is .iam. An assembly file links to a set of parametric 3D solid parts and/or sub-assembly of parametric 3D solid parts. It stores only the information on how the component parts are assembled together and information regarding the details of the parametric 3D solid parts is stored in the corresponding part files. Each time you open an assembly file, information from the part files is retrieved.

Presentation Files

To construct an exploded presentation of an assembly or animate the exploded presentation, you use a presentation file; the file extension is .ipn. A presentation file links to an assembly file. It stores the information on how the parts of the assembly are tweaked apart. Details regarding how the component parts are assembled are stored in the respective assembly file.

Drawing Files

To construct a 2D engineering drawing of a parametric 3D solid part, an assembly of 3D solid parts, and an exploded view of an assembly, you use a drawing file; the file extension is .idw. A drawing file links to a part file, an assembly file, or a presentation file. It stores the information about the 2D presentation of 3D objects.

File Icons

To depict the four kinds of files, different icons are used. See Figure 1–6.



Figure 1–6

Icons (from left to right) for part file, assembly file, presentation file, and drawing file

APPLICATION WINDOW

Now select the Cancel button of the Open dialog box. This brings you to the Inventor application window. (See Figure 1–7.)

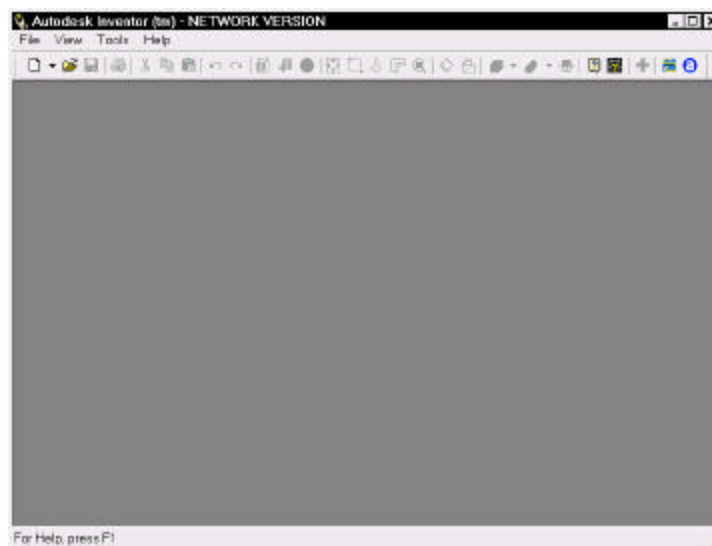


Figure 1–7

Application window

In the application window, there are five major areas. At the top of the window, there is the standard Windows title bar which displays the name of the application. Below the title bar, is a set of pull-down menus, and below the menus is the Standard toolbar (Figure 1–8). Below the toolbar is a graphics area, and at the bottom of the window, there is a status bar.



Figure 1–8
Standard toolbar

Starting A New File

Having decided what to do (construct a part, assembly, presentation, or drawing file), you start a new file by selecting New from the File pull-down menu. In the Open dialog box, select New from the What To Do panel. Then you will find three tabs: Default, English, and Metric. In each tab, there are a number of template files. The English tab has English templates; the Metric tab has metric templates (including BSI, DIN, GB, ISO, and JIS standards); and the Default tab has templates configured for the default measurement system you selected when you installed Inventor. Select the Default tab. You will find five template icons: Sheet Metal.ipt, Standard.iam, Standard.idw, Standard.ipn, and Standard.ipt.

New Part File

There are two part file templates, Sheet Metal.ipt and Standard.ipt. Figure 1–9 shows the application window for constructing a solid part or a sheet metal part.

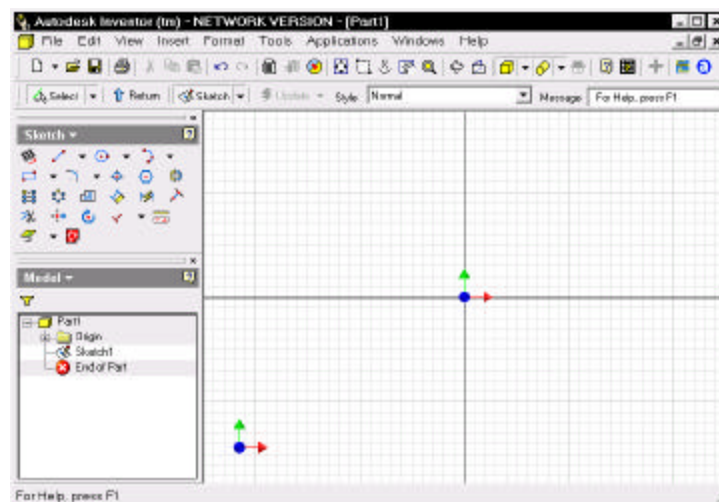


Figure 1–9
Application window for a 3D solid part file

New Assembly File

Select the Standard.iam template to construct an assembly of parts. Figure 1–10 shows the application window for constructing an assembly.

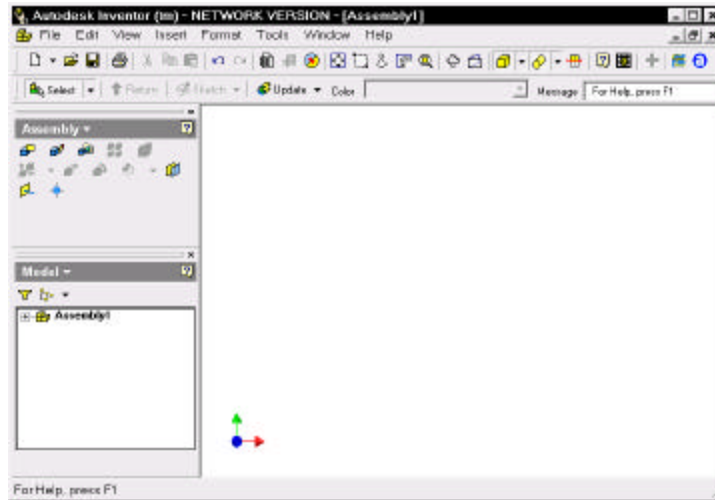


Figure 1–10
Application window for an assembly file

New Presentation File

Select the Standard.ipn template to construct a presentation of assemblies. Figure 1–11 shows the application window for constructing a presentation of an assembly.

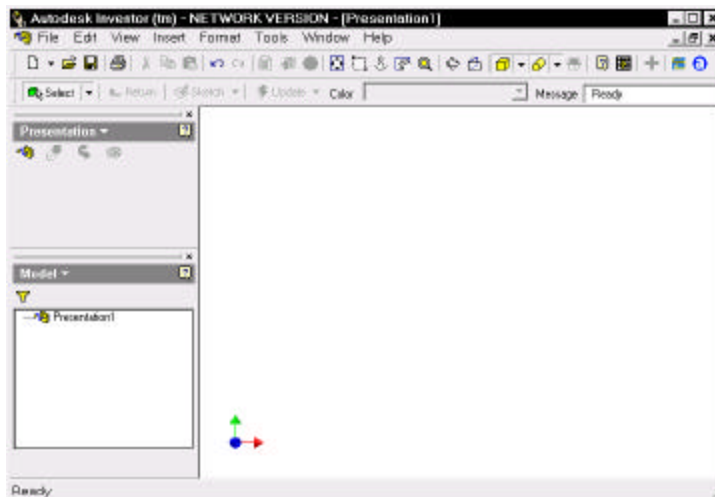


Figure 1–11
Application window for a presentation file

New Engineering Drawing File

Select the Standard.idw template to construct an engineering drawing. Figure 1–12 shows the application window for constructing an engineering drawing.

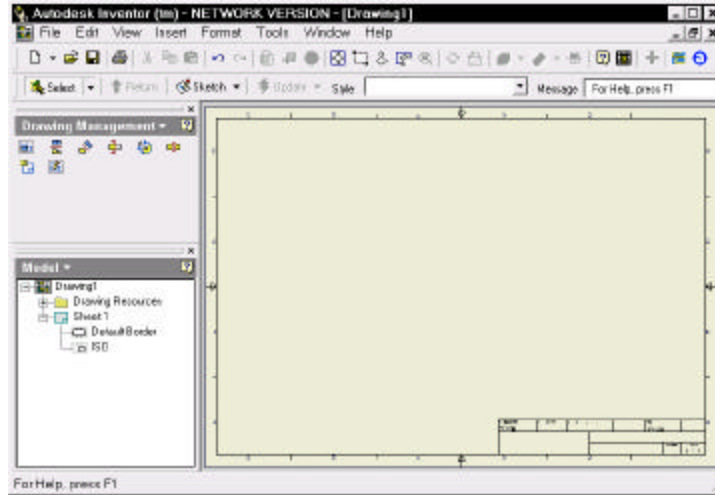


Figure 1–12
Application window for a drawing file

Panel Bar and Browser Bar

Common to the four application windows, there are three additional window areas: the panel bar and the browser bar. The panel bar has a number of palettes that enable you to access various design tools. The panel bar is context sensitive, so the tools available will vary according to current design context. In each palette of the panel bar, there are two modes available: general mode and expert mode. In general mode, there is text accompanying each icon to depict the command. In expert mode, text is not displayed until you place the cursor on the icon. Figure 1–13 shows the Sketch panel bar in general mode.

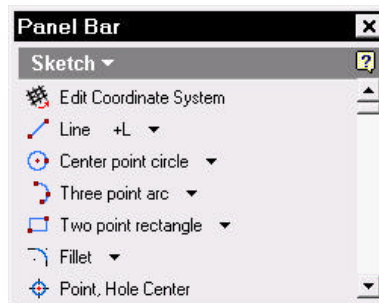


Figure 1–13
Sketch Panel bar

The browser bar shows a hierarchy of objects in the file. Figure 1–14 shows the browser bar for a part file. To increase the graphics area, you can close the panel bar and the browser bar.

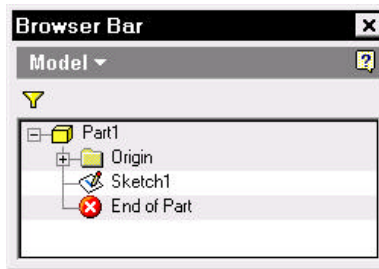


Figure 1–14
Browser bar

Right-Click Menus

Normally, your mouse has two buttons. You use the left button to select an object and use the right mouse button to activate a shortcut menu. Depending on the location of the mouse cursor and the kind of file you are working on, right-clicking the mouse will bring up different kinds of shortcut menus—they are context sensitive. You select an appropriate command from it. You will use the right-click button very often in the following chapters while you work through the tutorials. Figure 1–15 shows a right-click menu when you place your mouse on the graphics area of a part file.

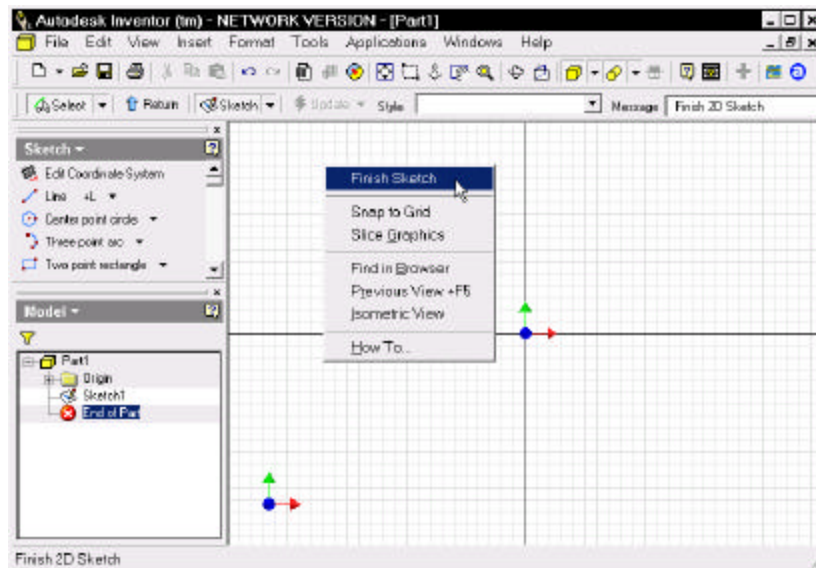


Figure 1–15
Context sensitive right-click menu