Introducing Autodesk Inventor® 2009 and Autodesk Inventor LT™ 2009

THOM TREMBLAY

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Introducing Autodesk Inventor® 2009 and Autodesk Inventor LT™ 2009
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Best regards,

Neil Edde
Vice President and Publisher
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Dedication

To Tony Fox, a true friend who is the epitome of what it means to be an Inventor.
I would first and foremost like to thank my wife Nancy for convincing me that I was capable of writing this book, then putting up with me while I did it. I hope that everyone who reads this has someone like her to lean on. I’d like to thank tech editor Steve Warren for all of his hard work and expertise, along with his willingness to put in the overtime. Thanks also to my good friends Mick Fears and Don Strimbu, who were the first to recognize that my passion for these technologies had other uses and encouraged me to find out all that I was capable of. I’m working on a new word to describe my gratitude for the Sybex team who endured my learning curve: Willem Knibbe, Jim Compton, Melissa Lopez, Eric Charbonneau, Laurel Ibey, Sandy Jaffe-Belanger, Janet Chang, Pete Gaughan, Liz Welch, and the others whose names that I don’t know but whose professionalism (and patience) was beyond compare. I also want to extend my sincere thanks to Lynn Allen, Alan Jacobs, and all of the Autodesk family who supported and encouraged me during this effort. Many thanks to Mr. Jay Tedeschi, a good friend whose fantastic Helicopter Rotor Head is featured on the front cover and is just one example of what can be done with Inventor and a creative mind.
About the Author

Thom Tremblay supports the Autodesk Worldwide Education group as a technical specialist on Inventor and AutoCAD Electrical. In his first seven years with Autodesk, Thom supported the Autodesk Inventor commercial sales group, working with customers to understand and improve their processes using Inventor and other technologies.

Thom’s professional experience includes working in the design of facilities, plastics, electronics, ship building, and commercial air conditioning. Early in his career he realized the benefits of using 3D geometry to create the 2D drawings that are so critical to create the products of the vast majority of companies around the world.
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In the spring of 1998 I was working for an Autodesk reseller at an event that we hosted. I was shown a preview of a yet-to-be-released technology that would eventually be known as Autodesk Inventor. It was simple and quick, and looked like it would be effortless to learn. I was hooked.

Ten years later Autodesk Inventor is the world’s largest-selling 3D mechanical design application. Its capabilities are branching into areas that until now were exclusive to applications that cost nearly ten times the price. With these capabilities comes the potential for complexity, but Inventor looks and behaves remarkably like it did when it was first released in the fall of 1999.

When the opportunity to write a book introducing people to Inventor came about, I was thrilled because I have seen how it has helped so many people. I immediately decided that I wanted to try to structure the book for the working designer and engineer who wants to learn Inventor but has very little time. I also wanted to change the order in which Inventor is usually taught, and the process for learning it, with the working professional in mind.

Traditionally, learning a 3D-based system begins with you constructing models, then learning how to assemble those models together, and finally (after many, many hours) creating a drawing. In my experience the people who would benefit most from Inventor are those who are spending much, if not most, of their time making 2D drawings. So why not begin the hands-on work with Inventor by working with 2D drawings and seeing just how easily they’re made and how powerful they are?

The next decision was to introduce these processes in stages. I wanted you to be able to spend time with the most fundamental tools and focus on getting comfortable with them before learning the more advanced tools in each of the main portions of the program. I also think moving more quickly through the fundamentals will expose the consistency in the Inventor approach and build your confidence and comfort level as you see that the dialog boxes and other elements for each tool have something in common with the tools you’ve already learned. In fact, one of the greatest challenges was paring down what to include, because an advanced feature is usually only a click or two away from the most basic tools.
Finally the use of the individual tools is also meant to be absorbed in stages. First I will try to walk you through the use of a particular tool; when you use the tool later you may be reminded of where to find it or a required step, and then later you will just be asked to use the tool as part of learning another topic. Hopefully this will build your confidence while allowing you to focus on new tools rather than ones you’ve used repeatedly.

**Who Should Buy This Book**

This book is not intended as a text on creating engineering drawings or to teach engineering or design philosophy. This book assumes that you are:

- A working professional with design and drafting experience
- Familiar with basic Microsoft Windows functions
- Familiar with drafting and design terminology

For example, this book will discuss how to create orthographic projections in a drawing but will not discuss what an orthographic projection is.

Given that, I don’t assume readers have 3D solid modeling or even experience with a computer-aided design (CAD) program. If this book encourages people who’ve never used 3D or even CAD to try Inventor, then I will consider it an unqualified success.

The exercises in this book will focus on how Autodesk Inventor and Autodesk Inventor LT can be used as a tool to carry out design work for a production environment.

**System Requirements**

The basic system requirements for Autodesk Inventor Suite and LT 2009 are as follows:

- Operating systems
  - Windows® XP Professional, Home, XP Professional x64 SP2
  - Windows® Vista (32-bit or 64-bit)

- Hardware
  - Intel © Pentium © 4 or AMD® Athlon® 64 or later. 2GHz or faster processor.
  - 512MB of RAM (minimum); >1GB recommended
  - >1.5GB free disk space for Inventor LT and >3GB for Inventor Suite with Content Center
  - Direct3D 9 or 10 Graphics support with >64MB of RAM
What’s Inside

Here is a preview of each chapter.

Chapter 1: Inventor Face to Face  This chapter presents the interface and working environment of Autodesk Inventor.

Chapter 2: Creating 2D Drawings from 3D Data  Creating production drawings is critical for most designers. This chapter introduces the primary tools used by Inventor to create 2D drawings.

Chapter 3: Introducing Part Modeling  The fundamentals of building 3D parts are the focus of this chapter. Understanding sketching and the application of parametric dimensions and constraining the movement of sketch elements are a big part of the exercises.

Chapter 4: Pulling It All Together with Assemblies*  Building assemblies out of individual parts is an important part of developing products using Autodesk Inventor. This chapter takes you through the process of making common assembly models.

Chapter 5: Standards and Styles  Creating parts and drawings consistently is important, and using the same layers, dimensions, and other styles makes it much easier to do.

Chapter 6: Advanced Annotation: Drawing Views and Detailing*  Continuing the topics introduced in Chapter 2, this chapter works with additional drawing view creation and annotation tools.

Chapter 7: Getting in Shape: Advanced Part Modeling  In this chapter you will work with tools not covered in Chapter 3. These tools will include special editing tools, advanced sketching techniques, and family parts.

Chapter 8: Advanced Assembly Tools*  Additional tools for building assemblies are just a part of this chapter. Learn about Inventor’s true design capabilities using Design Accelerators and families of assemblies.

Chapter 9: Introducing Sheet Metal Parts*  Tools are available for sheet metal parts. This chapter will take you through the basics of creating these specialized parts.

Chapter 10: Introducing Inventor Studio  Inventor Studio is an environment inside Inventor that allows you to create photo-realistic renderings and animations from 3D models.

* Topics covered include functionality not available in Autodesk Inventor LT 2009.
You will need to download data for some exercises from www.sybex.com/go/introducinginventor2009. The files will be contained in a zip file and downloading the data beforehand will save time.

**How to Contact the Author**

I appreciate your interest in this book and in Inventor. If you would like to share your feedback or stories about how this book may have helped, I would love to hear from you.

You can reach me at thom.tremblay@yahoo.com

Sybex strives to keep you supplied with the latest tools and information you need for your work. Please check the website at www.sybex.com, where we’ll post additional content and updates that supplement this book should the need arise. Enter *Introducing Autodesk Inventor* in the Search box (or type the book’s ISBN: 9780470375525), and click Go to get to the book’s update page.

Now let’s begin exploring Inventor, and hopefully you’ll have some fun along the way.
This chapter focuses on the Autodesk Inventor Suite’s 2009 interface and the fundamental tools for accessing files, working with the tools of Inventor, and getting additional information.

One of the greatest barriers that I’ve found to learning new software (or learning any new task) is the feeling that you have no control over the environment you are working in. In this chapter we’ll look at ways to modify the look and feel of Inventor so that you feel more comfortable and learn where things are. A thorough understanding of the interface may not seem exciting, but getting comfortable with the interface and knowing where to look for help is the foundation for everything else that you will learn in this book.

- Opening existing files
- Understanding Inventor’s Interface behavior
- Modifying the look and feel of Inventor
- Managing file locations
- Accessing the Help system
Learning to Use the Dialog Boxes

One thing that makes Inventor easy to use is a kind of “graphical language” common to all the dialog boxes. These are items that behave consistently wherever they appear. As you use Inventor, working with these items will become second nature. As you’re getting started, knowing what to look for will make it easy to understand what Inventor needs from you in order to accomplish your task.

Buttons

The following buttons and button states have the same effect no matter where you encounter them in Inventor’s interface:

- A button with a red arrow indicates that Inventor needs you to select something. Text may appear next to the arrow, identifying the type of input that Inventor is looking for.
- A button with a white arrow means that Inventor has been given the information that it needs.

  The OK button will be grayed out until Inventor has the necessary user input to execute an operation. Clicking OK initiates the command or function and closes the dialog box.

  The Apply button will also be grayed out until Inventor has the necessary user input to execute an operation. Clicking the Apply button initiates the command or function but it will not close the dialog box. This allows you to execute the function and start using it again immediately.

  The Cancel button closes the dialog box without executing any operation.

  The More button exposes additional options for a dialog box. Once those options become visible, the arrows then point to the left so you can hide the options again.

  Any button with an ellipsis after the name launches another dialog box or selection window when clicked.

Tabs

Another element of the common graphical language is the way dialog boxes are organized. Many dialog boxes have tabs across the top, with each tab offering additional options. Though most common functions are contained on the first tab, when you begin working with a new dialog box, it is worth taking a few moments to explore what options are on the other tabs. For example, in the Extrude dialog box illustrated here, the Shape tab offers the basic options to select the shape and define the distance it will be extruded, while the More tab offers options to apply taper or draft to the shape.
Context Menus
You can access a large number of Inventor’s tools by clicking your secondary mouse button—typically the right button—at different places on your screen. As in other Windows software, right-clicking displays a context menu of options that are relevant to what you’re doing at the time. For example, as you’ll learn in Chapter 5, right-clicking a drawing’s border in the Browser displays a menu that lets you delete the border or add a new one. In the exercises and examples in this book, I’ll often instruct you to right-click and select the next operation from a context menu.

The Open Dialog Box
Each time you start Inventor you will be presented with the Open dialog box (see Figure 1.1) to select the file(s) you want to work on.

Inventor LT users will see a slightly different Open dialog box. Inventor LT does not have assembly capabilities, so there are elements that are not necessary to have. It will still be beneficial for the LT user to understand the capabilities of Inventor Suite or Professional 2009 in case they use it in the future.

As in any contemporary software, this dialog box allows you to select a file or files to open in Inventor. If you’re accustomed to Microsoft Windows Explorer and some of its viewing options, this dialog box will seem familiar. Using it should be comfortable for you right away. There are several components to the dialog box, and it is important to understand what these parts are and what they will do for you.

It is possible to resize the dialog box by clicking and dragging the corners in order to allow easy viewing of the information it displays.
Shortcuts and the File List

In the upper left of the dialog box is an area with a list of shortcuts to Frequently Used Subfolders (Figure 1.2). You can customize this pane to create shortcuts to folders that you’d like to access quickly. You can even set up subreferences and have a structure that replicates the folder structure on your hard drive.

Centered in the dialog box and making up the bulk of it is the File list, where the files are displayed. What files are listed is controlled by the File Of Type option, described shortly. You can open a file (or files) from here by selecting the filename(s) and clicking OK or by double-clicking on the filename.

At the top of the dialog box is the Look In field. This displays the name of the folder whose files are currently displayed below it in the File list. The arrow to the right allows you cascade the folder structure or to begin browsing for other folders.

Navigation Controls

To the right of the Look In field are four icons that allow you to navigate easily and to control how you view the files that you’re looking for.

These tools share icons and functions with many standard Windows icons and tools. You will find commonality in the controls between Inventor and many Microsoft applications. This is done so that you don’t have to learn every aspect of the user interface from scratch.

Go To Last Folder Visited The first button has an arrow pointing left. This button allows you to navigate back to the previous folder(s) you were browsing in. It works on the same principle as the Back button in a web browser. When you've just begun a session, the arrow will be grayed out, as you don't have any browsing history to recall.

Up One Level The next icon looks like a folder with a green arrow pointing up. This takes you up a level in your folder structure from wherever you are currently browsing.

Create New Folder The third icon allows you to create a new folder in the folder that you are currently browsing in.

The View Menu The icon on the right is a flyout tool that allows you to change the way the files that you’re browsing will be displayed. Depending on the operating system you’re using, you will see different options ranging from a detailed listing of dates and file size to thumbnail previews of the files in the display area. In Figure 1.3 you can see the same folder as in Figure 1.1 being browsed with the Thumbnail display option.
File Display Options

Immediately below the File list are three selection pull-down lists that control the file display options.

File Name  This pull-down displays the full name of the selected file(s). If you click the arrow to the right, it opens a list of recently open files.

Files Of Type  This option is very important. Clicking the arrow to the right lets you choose from a list of file types that Inventor can open. It’s important to filter the file types displayed because of the broad array of types.

Project File  This flyout allows you to select from a list of project files that have been used in the past. The active or current project file is shown any time the Open dialog box is brought up. To the right of the pull-down is a button marked Projects, which launches the Project File editor, which in turn allows you to select project files that have not been used previously, edit existing project files, or create a new project file. We’ll take a look at the Project File dialog box later in this chapter in the Project Files section.

To the left of File Display Options is the File Preview pane. As you select a file in the File list, a preview of that file appears in this area. Not all files have a preview to display.

At the lower left of the Open dialog box you’ll find three icons under the heading Quick Launch.
At least one of these icons will not be available at any given time. If you’re in the Open dialog box, the first icon will be available; it switches you from Open to the New dialog box. The middle icon switches you back to Open from New. The third icon is for opening files from the Vault, a great data management system that I highly recommend installing. (The Vault comes with Inventor so there’s no additional cost but there are great benefits that I will talk about briefly in Appendix C.) This icon will be available only if the project file that is active has the Vault enabled.

In the lower right you’ll see the Find button. Clicking it displays a Find tool dialog box (Figure 1.4) that can execute simple or complex searches. You can search for file properties, creation dates, or strings of text, and you can even save your searches to be reused in the future.

**Other Controls**

Three more options complete the Open dialog box:

**Options** Available only when you import, export, or open a file that can have additional settings applied to it. For example, if you want to export a DWG file for use with AutoCAD, you can select which version of AutoCAD can open the file, back to AutoCAD 2000.

**Open** Executes the opening of the selected file or files. You can open multiple files at the same time by holding the Ctrl key to select multiple individual files or by holding Shift to click a range of files. You can also open multiple files by dragging a file or files from Windows Explorer onto the title bar of Autodesk Inventor.

**Cancel** Exits the attempt to open a file and returns you to Inventor.

**Opening a File**

Now that you have had an overview of the parts and functions of the Open dialog box, let’s put what you’ve learned to use. (Some options will not be available to Inventor LT users; again, LT cannot work with assemblies.)

1. If the Samples project isn’t displayed as the active project, use the pull-down list to select it. It should be on the short Project File list. If it doesn’t appear on the list, open the Project File dialog box as described earlier, select Samples from the list, and click Open. This should set that project to be active and return you to the Open dialog box.
2. Use the Frequently Used Subfolders list to find the Assemblies\Engine MKII\Engine MKII.iam assembly file. Inventor LT users should select a part file of their choosing to review changes. You can find samples files to choose from at C:\Program Files\Autodesk\Inventor LT Technology Preview\Samples.

3. Once you have found the file, you can select it with a single mouse click and click OK, or you can double-click on the file in the window. Once the file is open, you should see something like Figure 1.5.

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**The New File Dialog Box**

The New File dialog box (Figure 1.6) is much simpler than the Open dialog box. Like the Open dialog box, it has a Quick Launch section that allows you to switch to the Open dialog box, and where you can set the active project file.

Every new drawing you create in Inventor is based on a template, which provides information such as borers, title blocks, layer colors, and the standard dimension style. These templates can be customized, and Inventor comes with a sizable selection to give you a head start. It is also possible to convert existing AutoCAD drawings to Inventor templates.

In the New File dialog box, Inventor’s collection of built-in templates is categorized in tabs across the top. There are templates for Default, English, and Metric measurements.
Navigating Inventor’s User Interface

With Inventor open and a file loaded, it is time to get familiar with the component of Inventor that you will use every time you start the program; the graphical user interface (GUI). Figure 1.7 shows the main components.
If you're used to the look and feel of Microsoft Word (prior to 2007) and Microsoft Internet Explorer, you should find a lot of similarities to many of the standard tools in Inventor. If you are experienced with recent versions of AutoCAD, you will find a lot of similarities with the drawing and sketching tool icons in Inventor.

Across the top of the Inventor application is the title bar. It will remind you that you are in Autodesk Inventor and tell you what version you are using. It will also tell you the name of the file that you are currently editing if that file is in its maximized view. If the Inventor file is in a floating window, then each window's title bar will display the filename.

Just below that is the menu bar, a collection of tools is organized in pull-down menus. These menus contain valuable tools for everything from an alternative way of opening files to measuring geometry on the screen. You can even launch a website containing components manufactured by other Inventor users that you can download into your designs. The names, content, and number of these menus will vary depending on whether you have a file open and what type of file it is.

Over the course of the book we will examine some of the tools in the menu bar pull-downs as needed. A majority of the tools in the menu bar establish how you are working with Inventor rather than containing the tools that you would use for modeling or drawing.

Immediately below the menu bar is the Inventor Standard toolbar. This is where a lot of the basic file and view manipulation tools can be accessed easily. The toolbar is a collection of groups of different types of tools. The tools available depend on the type of file you're editing at the time. Most of the differences will be found on the right end of the toolbar. The first three groups of tools are consistent regardless of the type of file you're editing.

Since you have the assembly loaded, let’s use it to explore how you will interact with Inventor and take a closer look at some of the elements of the interface that we just touched on.

Let’s begin with an in-depth look at the Inventor Standard toolbar. Most of the other elements of the interface will be used as part of exercises, but the Standard toolbar contains tools that are used to control and better understand the model that you’re creating. These tools do not create geometry, but they make it easy to do so.
On the left you may recognize the standard icons for New, Open, and Save. The third icon may not be immediately recognized; it is Open From Vault. This is the same tool that you saw in the overview of the Open dialog box.

The next section has to do with working in the context of parts and assemblies. The first two icons are our old friends Undo and Redo. If you make a mistake, Inventor will allow up to 30 steps of Undo and Redo. A great feature is that changes to the model view (zooming, panning, etc.) do not use Undo steps. You can even undo the creation or opening of a file.

Immediately to the right of those tools is the selection Filters list. Filters are a tool for focusing or streamlining selections. They can limit or enhance the selection of certain types of entities in parts, assemblies, or drawings. The use of filters is a great thing to learn and explore. Many experienced Inventor users are missing out by not becoming more comfortable with them.

You will use the Return button frequently. This tool moves you from one editing state to the one above it. Its importance will become clear as we start working in Inventor.

Several toolbar buttons have a down arrow next to them. This indicates that an additional option or options are under the primary command. For example, the Sketch icon, which allows you to create or edit sketches, has a 3D Sketch button under it.

If Inventor is unsure that what it is displaying is the most current information, the Update button becomes available and allows you to update the data that is on the screen.

The third portion controls how you look at the file you are editing. Many of the icons will be familiar. They are commonly used tools, so take a moment to practice using them in context. Begin on the left and work your way across.

The first icon is Zoom All. No matter what your point of view, clicking this icon frames your model evenly in the Design window, which is where you are currently seeing the engine assembly.

Zoom Window allows you to zoom in on a specific area by creating a “window” frame around the area that you want larger. To create the frame, you select the tool, click where you want one corner, and while continuing to hold the button down, drag the size of the frame. When you’ve encompassed the area you want to make larger, release the mouse button. Figure 1.8 shows the zoom area being framed, and Figure 1.9 shows the result. Try enlarging your view of a portion of the assembly using the Zoom Window tool. You can also access Zoom Window by pressing the Z key on your keyboard.