

Chapter 2

Data and Projects

In many design and manufacturing environments, teamwork is a way of life—an essential part of getting a product to market quickly. Concurrent design among multiple team members requires coordination, discipline, and organization. In other situations a designer might work primarily as a stand-alone user, collaborating with others but generally creating and accessing files as a single user. In either case, the effort invested in setting up an efficient file management system will be worth it in the time you save while designing parts, and the safeguards it provides against rework and downstream errors for the designer. When working as part of a design team, the value increases exponentially.

In this chapter, you will learn how to:

- ◆ Understand how project search paths work
- ◆ Set up library and Content Center paths
- ◆ Create and configure a project file
- ◆ Determine the best project type for you

What Is an Inventor Project?

You can think of project files in Inventor simply as configuration files that tell Inventor where to look for component files when working with assemblies and drawings. For instance, an Inventor assembly file is essentially an empty “bucket” into which parts (and subassemblies) are placed and assembled. Therefore, the assembly file contains only the file path references for the components it is composed of and the information about how those components are assembled. As a result, the location of referenced files is a key issue. If, when an assembly is opened, referenced files cannot be found at the search path recorded in the assembly file, a manual file resolution process is activated. This happens most often when component files are renamed or moved outside the search cone established in the project file.

A NOTE TO INVENTOR LT USERS

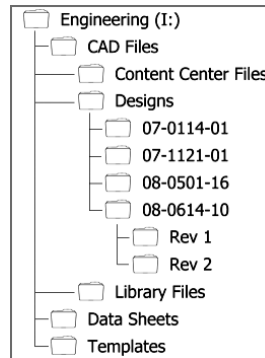
If you are using Inventor LT, you should be aware that it doesn't make use of project files and therefore instruction concerning Inventor project files does not apply.

Project Files and Search Paths

Project files are often referred to as IPJ files because `.ipj` is the extension for project filenames. You can create a project file anywhere it makes sense to do so, and Inventor will look at that location and lower in the directory structure for the files in your design. Take a moment and study the file structure shown in Figure 2.1.

Figure 2.1 shows a typical job-based folder structure, where all files are located on the Engineering (I:) drive. Engineering contains three subfolders: CAD Files, Data Sheets, and Templates. In the CAD Files folder are three more folders: Content Center Files, Designs, and Library Files. The Designs folder contains a folder for each job (named using the job number) and subfolders containing revisions. So where would you create an Inventor project file? There are two basic solutions: create multiple IPJ files for each new job or create a single all-encompassing IPJ file for the entire engineering drive. Which method you should use depends largely on the way your engineering department operates.

FIGURE 2.1
A job-based folder structure



JOB-BASED IPJ SETUPS

You could create one project file for each of the four jobs. You would have a file named `07-0114-01.ipj` in the `07-0114-01` folder, one named `07-1121-01.ipj` in the `07-1121-01` folder, and so on. This strategy can work fine if you typically work on one project at a time and then “close” the project upon completion. In Inventor, you would simply switch to the specific project file that matches the job number for the job you are working on (for example, `07-0114-01`), and because the IPJ file is stored in that folder, Inventor will only search for design files in its *workspace*. The workspace is defined as the folder containing the IPJ folder and everything below it. It may help to think of a workspace as a search cone, starting at the IPJ and spreading out from that point.

This job-based approach is fairly intuitive and is what people generally think of when they see the term *project file*; one IPJ file for each job/project. This is a common approach when a job has a long development cycle and designs are very specific to that job.

But what happens if you want to use a part that was created for job `07-0114-01` in job `08-0614-10`? You could place the part into the `08-0614-10` assembly, but the next time you opened that assembly, Inventor would not be able to find it because it exists outside of the `08-0614-10` workspace. If you were to move a part file from the `07-0114-01` folder into the `08-0614-10` folder,

Inventor would not find it while you were working on job 07-0114-01 because it would now be outside of its workspace. Likewise, if you moved the file up to the Designs folder, or the CAD Files folder, or to (almost) any location that is not next to or below the 07-0114-01 .ipj file, Inventor would not find it as long as you are working with the 07-0114-01 .ipj project file. If you copy the file to the 08-0614-10 folder, then you have two versions of it and it becomes difficult to track changes because you need to update both copies to keep everything up-to-date.

The solution would be to configure the IPJ file to include a *library*. When a folder is configured to be used as a library in an IPJ file, Inventor sees all of the files in that folder (and its subfolders) as read-only. This protects commonly used files from being accidentally changed and upsetting all of the many designs in which they may be used.

To solve the issue of the commonly used part in this example, you could configure each of your IPJ files to use the folder named Library Files as a library of approved, read-only parts to be used across multiple jobs. Whenever you open an assembly, Inventor first looks in the library path for the parts and then looks at the workspace. So to convert a part created as part of the 07-0114-01 job into a library part, you would follow these steps:

1. Copy the file to the library folder.
2. If the original file has a job-specific name, rename the copy according to a defined library nomenclature.
3. Open the assembly (or assemblies) that use the original part.
4. Use the Replace Components tool to replace the original part with the library part in the assembly.
5. Save and close the assembly.
6. Delete the original part so that no duplicate is present.

SINGLE IPJ SETUPS

Using the same folder structure as shown in Figure 2.1, you could use a single all-encompassing IPJ file and set it in the Designs folder. By doing so, you would be setting the workspace at that level. This configures Inventor's search paths to look for files in the Designs folder and everything below it. Essentially, you have expanded the search cone by moving it up a level compared to the job-based setup. Now if you need to use a part that resides in the 07-0114-01 folder in the 08-0614-10 assembly, you can do so and Inventor will be able to find it, without requiring it to be in a library folder.

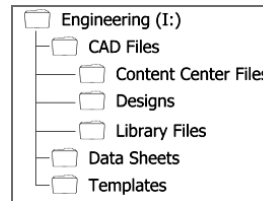
Of course, you can still use library folders when using the single IPJ file approach, and in fact it is generally recommended that you do convert common parts to library parts when they are being used in many different designs. Because folders configured as libraries in the IPJ file are handled as read-only, this protects them from accidental modifications.

One major caveat to using just a single IPJ file is that in order to prevent the possibility of the wrong part being loaded in an assembly, it is important for every part located in the search path to have a unique name. If Inventor finds two files named BasePlate01, it will either use the first one it finds or stop and make you decide which one to use. In either case, you should consider a nomenclature that references the job number, date, or other unique identifier in the name.

ITEM-BASED SETUPS

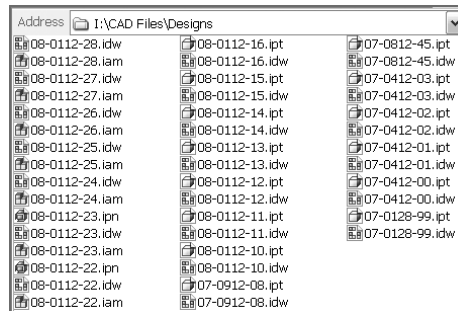
If your company uses an item-based file management setup and tracks each part you create or purchase as an item, then you are probably not concerned with job numbers as much as you are about part numbers. Most likely you will want to employ a single IPJ file setup as described previously and again place the IPJ file in the Designs folder. Additionally, your file structure may be a bit more flat and look like Figure 2.2, where the Designs folder has no subfolders.

FIGURE 2.2
An item-based folder structure



In this flatter structure, you can simplify the folder structure and drop all files into the Designs folder as shown in Figure 2.3.

FIGURE 2.3
A simplified folder structure



Of course, you could also still populate the Designs folder with subfolders named by product line, by top-level item, or for each job, just as it's done in Figure 2.1. Typically, it is best to set up the IPJ file to accommodate your current file management system. However, if your current system is a mess or is simply no longer a good fit for your company, you might take the time to reorganize and plan out a good system and set up Inventor accordingly.

Library Folders

As described previously, library folders contain existing, shared components. Library folders are useful repositories for purchased parts such as fasteners, clamps, motors, and connectors as well as any other common, standard components. Library folder paths are defined in the IPJ file. Once the IPJ file is configured, all components stored in a folder designated as a library file are considered to be read-only by Inventor. This prevents the component from being unintentionally edited or from being revised without appropriate approvals. For example, before you modify a design that was completed as part of another job, it's important to determine where else that part was used. The goal is to ensure that the changes you plan will not render the part unusable for other designs.

Library folders should be located outside the main IPJ workspace path. In the job-based directory structure example shown in Figure 2.1, the Library folder is on the same directory level as the Designs folder and therefore outside the workspace search path. Library folders can be located anywhere outside the primary project data path, even on different drives or mapped servers. You should note that if you set up a library path in the IPJ file to a folder that does not exist, Inventor will create the folder as specified in the path. A good way to set up libraries is to set the path, let Inventor create the folder so that you know it's in the right place, and then populate the folder with the library files.

LIBRARY EDITOR IPJ FILES

So if folders configured as libraries are configured as read-only in the IPJ file, how are controlled, purposeful, revisions carried out on library files? The answer is to create a regular IPJ file configured to look at the library folder as a standard folder. For instance, you might create an IPJ file in the Library Files folder and assign it no library path. You would then switch to this IPJ file only when doing library maintenance. Because this IPJ has no library path called out, the files are not handled as read-only. Often in a large engineering department only a couple of people have access to the library editor project file. When other team members see a need to change a library file, they would submit a change order and the designated person(s) would then make the change.

Content Center Files

In the previous figures you may have noticed a folder called *Content Center Files*. This is a special kind of library that stores component files generated by the Content Center. The path to this folder is specified in the IPJ file, much as a library file is.

It is important to understand what Content Center is and how it works. The Content Center libraries are collections of table data containing the definitions for how to create over 800,000 standard parts and features. This database is managed by the *Desktop Content* settings or the *Autodesk Data Management Server (ADMS)*. Once you've installed the content libraries, you can use the content in your designs. To do this, choose a component from the database to place into your design, typically by using the Place From Content Center button in the assembly. It is at this point that the Content Center part file is created. Up until this point, the part existed only as a definition in the database table.

In your IPJ, you need to specify a Content Center file store location so that Inventor knows where to save the file, and where to find it next time. The file store folder will include additional subfolders where Content Center files will be stored once they are used in your designs. These additional folders are created automatically as parts are created. The next time a part is specified from the Content Center libraries, Inventor first searches the Content Center file store directories and then creates the part from the database only if the part file does not already exist in the file store location. It is required that the Content Center file store location be outside the main project data path. From this discussion of libraries, you can see that high importance is placed on planning the correct part locations and workflow.

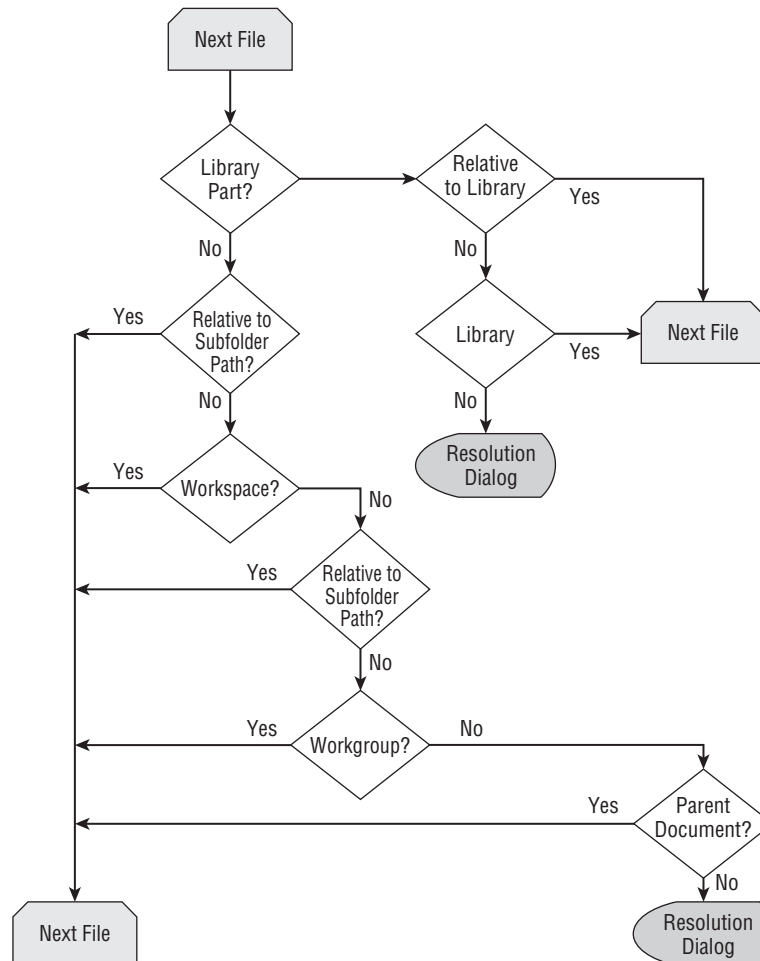
How Search Paths and Project Files Are Used

Inventor's IPJ files are easy to create and use, provided you understand how Inventor uses them. An Inventor project file is a configuration file that lists the locations and functions of each search path. Inventor uses these definitions to resolve file links and locate the files needed for the parts

and assemblies on which you want to work. Figure 2.4 shows how Inventor loads assemblies and parts inside an assembly file.

When opening an assembly file, Inventor finds files by searching for the first file to be located within the assembly file. Inventor first looks in the library folders for that file. Next, Inventor searches in the local workspace for the file. When a file is not found in any of the referenced folders, Inventor launches a manual file resolution dialog box offering you the opportunity to browse and point to the file.

FIGURE 2.4
Inventor file resolution protocol



Exploring Project File Types

As mentioned previously, file management in Inventor is handled through the use of a project file (*.ipj). A *project file* is simply a configuration file set up and used to control how Inventor creates and resolves file links, where you edit files, how many old versions of the files to keep,

and how Content Center files are stored and used. In the early days of Inventor, Autodesk offered two basic project types: single-user projects and multiuser projects. At this point, the Vault project has replaced the earlier multiuser project types.

Unless you have installed Vault, you have only one project type to choose from by default, the single-user project. The term *single-user* could be considered a misnomer because this project is widely used by one-man shops and multiseat design departments alike. *Single-user* does not mean that only one user can access the files in the project, as it might suggest; instead it refers to the fact that there are no means of preventing files from being accessed for editing while another user is already editing the file. This can create a last-man-to-save-wins situation if care is not taken.

SINGLE-USER PROJECTS AMONG MULTIPLE USERS

What happens when two users access the same file in a single-user project? Typically this is first noticed when one of the two tries to save the file. Inventor notifies the person trying to save that they are not working with the most current version and gives the other user's name (depending upon network setup) so the first user knows what is going on. Inventor instructs the user that they must save the file using a different name to prevent using the changes made.

Typically, at this point a conversation takes place to determine how to proceed. If it is decided that the first person is the one who needs to save changes, then the file this person was working on is saved using another name, the original file is deleted (or renamed as a reserve), and the other file is renamed to replace the original. In this way, the changes that were made to the original file have been preserved.

Although this may seem like a terrible hassle, there are many design departments that use single-user projects in a team setting effectively, and only rarely run into this situation. More than likely you already have an idea of how often you and your colleagues handle the same files at the same time. But if you try to use single-user projects and find this situation happens fairly often, you should consider a true multiuser project.

Many design departments use single-user projects effectively in collaborative environments because of workflows that lend themselves to this type of project; others make it work by simply maintaining good communication among the design team. For collaborative environments that require some safeguard against situations in which users could potentially save over one another's work, using a *multiuser* project (Vault project) is recommended.

Vault is a data management application that, as the name implies, locks down files for their protection. Once a file is in Vault, it is checked out by a user to be edited. Vault typically resides on a file server where the entire design team can access it. When the file is checked out of the Vault server, it is placed on the user's local machine for editing. The next user who comes along and attempts to access that file can access only a read-only version. Once the first user is finished editing, the file is checked back into Vault and automatically versioned.

It is also important to note that Inventor installs with a *default* project setup. The default project is typically not used for production work because it is not fully configurable and will almost always lead to file resolution issues because it has no defined search path.

Creating the Project File

In Inventor, two project file configurations are available:

Single-user A single-user project allows you to work in a file structure that is wholly contained on your system or on a network server location. This project file type is the simplest to create and works well when users are not working on the same design concurrently.

Vault Autodesk Vault is an easy-to-use data management tool that integrates work created with Inventor, Inventor Professional, AutoCAD Mechanical, and AutoCAD Electrical. It includes features that allow design teams to track work in progress and maintain version control in a multiuser environment. Design reuse is facilitated by consolidating product information and storing it in one place. Vault is a SQL database environment. A subset of the SQL environment exists in all current Windows operating systems (back to Windows 2000). Vault installs separately from Inventor. The Vault installation checks to make sure that your system is compatible and that auxiliary programs required for operation are installed. Vault is included with all versions of AutoCAD and Inventor.

Now, which type of project is best for you?

One or more designers can use projects using Vault. Single-user projects are most commonly used when there is a single seat of Inventor in the company or when only one designer works on a particular job more or less exclusively.

Multiuser Vault projects rely on a Microsoft SQL Server environment. That can be as simple as the Autodesk data management server, which supports up to 10 users with the default Microsoft SQL Server Express database. If you have a larger workgroup or require a higher capacity, a full version of Microsoft SQL Server is recommended. In addition, a workspace folder located on the individual user's system is required. Data servers should be on a separate server with rapid data access hard drives dedicated to the engineering department's use.

Inventor 9 and earlier allowed you to create shared and semi-isolated project file types. If you are currently using either of these legacy project types, you should consider moving to Vault since Autodesk may not support the legacy types in future versions.

CREATING A GOOD DATA MANAGEMENT PLAN

A good data management plan is the key to using Inventor projects successfully. Using Vault will not resolve a poor project file or data management strategy.

One part of a successful Inventor deployment is the hardware and network on which the software will run. It is important that the engineering group has buy-in by the IT group. You will need to discuss several issues with this group, including hardware for servers and workstations, the network setup (100BaseT or Gigabit), mapped network drives, and user permissions. A good server can be the difference between success and failure in your rollout.

Although you do need to think about your file structure, don't obsess over it. Most likely you will end up changing the structure at least a few times before you settle on a final structure. Keep an open mind, and realize that if you have five people in a room discussing file structures, you'll end up with five different ideas. Again, involve IT in your discussions.

Finally, you should designate one person in engineering to be the engineering administrator. This person needs to have administrative privileges on the engineering server or network share. IT may resist, but you need to keep pushing. This is important because you will need the ability to easily create, delete, and move files and folders without having to submit a help-desk ticket. Nothing will slow down a design process faster than having to wait for IT to make a simple change. Explain this need to your IT administrator, and most likely they will understand.

Creating Single-User Projects

**CERT
OBJECTIVE**

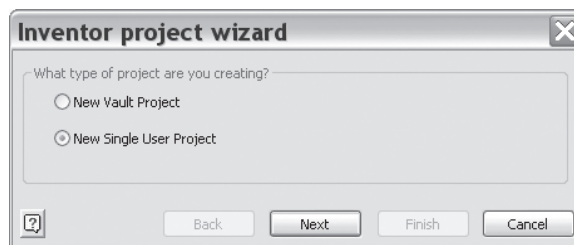
Probably the best way to learn about projects is to create a “test” single-user project. With single-user projects, you can open, edit, and save files without checking the files in or out. In the following sections, you will investigate the single-user file project mode. Once you gain an understanding of single-user projects, you will be ready to investigate the other project file types. To create a test project, you will use the Inventor Project Wizard.

THE INVENTOR PROJECT WIZARD

To get the most out of this exercise, open your version of Inventor, ensure that you have closed all the open files, and then access the Inventor Project Wizard by clicking the Projects button on the Get Started tab. Then follow these steps:

1. In the Projects dialog box, click the New button at the bottom.
2. If you have installed Vault, you will see two options in the Inventor Project Wizard, as shown in Figure 2.5. If not, you will see only one option. In either case, select New Single User Project, and then click Next.

FIGURE 2.5
Creating a single-user project

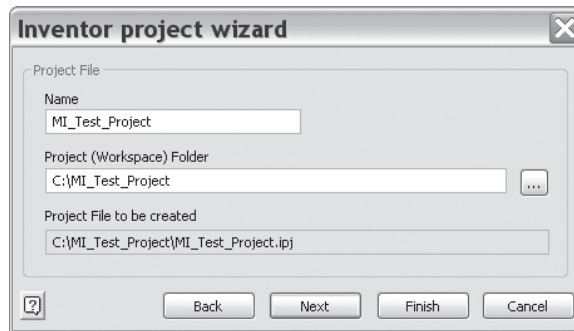


3. Enter `MI_Test_Project` in the Name input box.
4. Enter `C:\ MI_Test_Project` in the Project (Workspace) Folder input box.

Figure 2.6 shows a Project File page specifying the project.

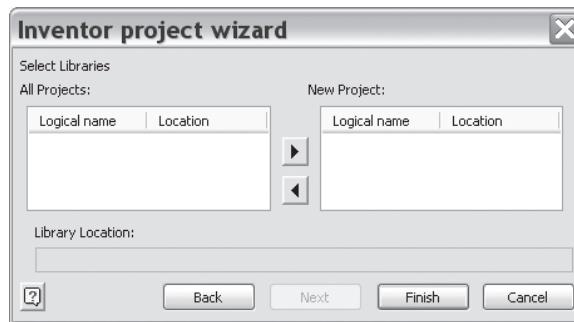
5. Click Next to advance to the next page of the wizard.

FIGURE 2.6
The Project File
page filled in



6. If you already created a folder for your library files and used those libraries in a previous project, their locations will appear on the Select Libraries page, shown in Figure 2.7. When creating a new project, you can choose to include some, all, or none of the defined library locations. Click the Finish button to include no libraries at this point.

FIGURE 2.7
Select Libraries
page



7. Click OK in the message box informing you that the project path entered does not yet exist.
8. Click Finish to create and save your new project file. The newly created project file link will appear in the list in the Projects dialog box.

SWITCHING AND EDITING PROJECTS

Only one project can be active at a time. To switch projects, you must first close all files that are open in Inventor. You cannot edit the file paths of the active project, but you can edit items such as Content Center libraries that are included. You can edit anything in an inactive project.

**CERT
OBJECTIVE**

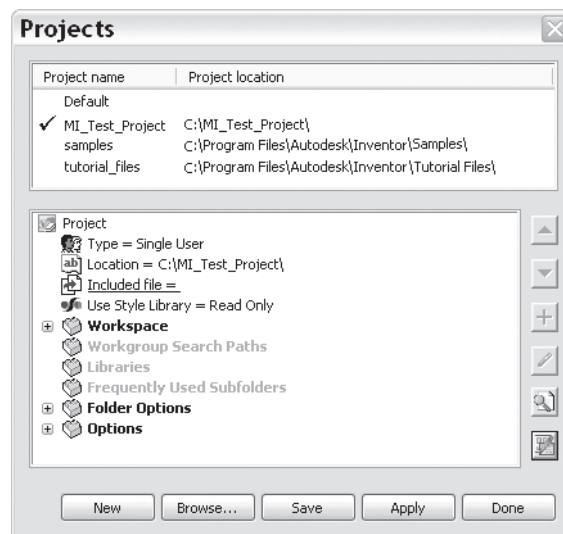
THE PROJECTS DIALOG BOX

Now that you have created your sample project file, you'll explore the options and settings available for your new project. To activate and use your new project, highlight the new project and click Apply. You can also activate or select a new project link by double-clicking the project link. Notice there is a check mark next to the project name `MI_Test_Project`, indicating that the project is now active, as shown in Figure 2.8. Double-click the part called `Face_Plate_mi_1` to set it active for edits.

In the lower pane, you can view and access parameter settings for the following:

- ◆ The project type
- ◆ The project location
- ◆ Optional included project file
- ◆ Style library options
- ◆ The workspace path
- ◆ Optional workgroup paths
- ◆ Libraries you want to use
- ◆ Frequently used subfolders
- ◆ Folder options
- ◆ Other project options

FIGURE 2.8
Projects dialog box



Right-click a parameter group to view the settings available within that group. Within the Project group, you can change the project type, view the project location, and include other project files. Project types were discussed earlier in this chapter. The project location is a read-only parameter. Included files deserve some additional discussion because the Included File parameter allows you to apply a master project to your current project; this setting, as well as the other project settings, are discussed in the coming pages.

Included Files

Although it's not required, you can include an existing project in the configuration of the current project by right-clicking Included File. The properties and settings in the project file that you attach override the settings in the current project file. This is useful for restricting and controlling a user's ability to change the project file. Also, if you frequently create new project files, you might consider creating a master project file that contains library locations and other settings you commonly use and then include the master project file in each new project file.

Workspace

For single-user projects, the workspace is defined by the location of the project file (*.ipj). For Vault projects, the workspace is defined on the workstation and is configured to match the Working Folder setting in the Vault settings. The workspace is the folder that files are copied to when they are checked out. The Workspace folder may include any number of subfolders as required for your file management needs.

Workgroup

The workgroup search path specifies a location outside the current project file paths where Inventor can search for existing files that are not included in a library. A workgroup is specified when the project is created. Each single-user project should have no more than one workgroup. Using a workgroup path is not required and is not a common configuration to make.

Style Library

Inventor uses styles to specify dimensions, text, colors, materials, and other properties. This is similar to styles used in AutoCAD. However, Inventor allows you to store styles locally within the templates or in an external style library that may be used with any project file.

The Use Style Library function in projects specifies whether the project uses only local styles, local styles and the style library, or just local styles and a read-only version of the style library. The read-only style library is recommended for projects that have multiple users. With multiple users, changing or editing the style library on the fly can cause downstream problems. To change the Use Style Library parameter, right-click and select the new setting.

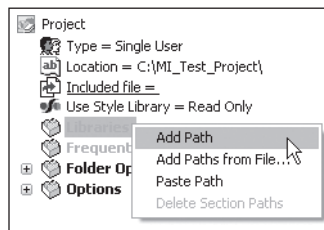
Remember that for your projects, you can right-click to select another option when it is appropriate. Click Yes if you want to be able to edit styles in this project. Click Read-Only (the default) if you want to access style libraries and local styles without enabling style-editing capabilities. Click No if you want to use only the styles located within the current file and project template. You can find more on using style libraries toward the end of the chapter.

Library Options

Next on the list are libraries. Library folders are located outside the project file path. They may be located anywhere on your system or on your server. If you are sharing library files, it is recommended that you place them on your server in a commonly accessed location. The contents of directories specified as libraries in the project file are considered read-only by Inventor.

In your newly created project file, you have not added any library folders. If at any time you want to add library folders, you can do so by right-clicking Libraries and choosing Add Path, Add Paths From File, or Paste Path, as shown in Figure 2.9.

FIGURE 2.9
Adding library paths by right-clicking



You can manually add a path, either by browsing or by typing a new file location. Be sure to give the library a descriptive name that identifies the contents of the file location. Add Paths From File permits extracting library paths from another project file. Paste Path allows the user to copy and paste. Once you have specified library paths, the Delete Section Paths option becomes available, and you can remove paths not needed by the project. Deleting unused library paths reduces search and resolution time.

Shortcuts to Frequently Used Files

Frequently used subfolders are similar to the bookmarks you can set in Internet Explorer. The subfolders must already be nested within the current project workspace, workgroup, or library. Adding frequently used subfolders to your project provides navigation links in your open, save, and place dialog boxes so you can quickly navigate to those locations. In the Mastering Inventor 2012 project that you will use throughout the rest of this book, the chapter folders have been added as frequently used subfolders.

Folder Options

The Folder Options setting allows your project to access other file locations than are specified on the Files tab of the Application Options dialog box. Keep in mind that you may have to close and reopen Inventor in order to reinitialize the optional project file locations. You can use this option to specify different default locations for templates, design data, styles, and Content Center files. When the locations are set to the defaults, then the location/storage of the files is specified on the Files tab of the Application Options dialog box. Right-click any of the options entries to change the storage and access location. You can find more information about design data, styles, and templates toward the end of this chapter, and more about Content Center in Chapter 7.

Project Options

Expand the Options heading to show the global defaults for the selected project. The Options settings in a project determine file management functions; right-click an option to edit it.

Versioning and Backup

Use the Options settings to determine how many old versions or backup copies of each file to save. The Old Versions To Keep On Save option specifies the number of versions to store in the Old Versions folder for each file saved. The first time a file is saved in a project, an Old Versions folder for that file is created. When the file is saved, the prior version is moved automatically to the file's Old Versions folder. After the number of old versions reaches the maximum in the setting, the oldest version is deleted when a newer version is moved into the folder.

If you are familiar with AutoCAD, you may expect Inventor versioning to be similar to AutoCAD's backup scheme. AutoCAD creates a *.bak file saved in the same folder as the design. Inventor saves the backup files in a separate Old Versions folder. All versions located in the Old Versions folder have the same name and extension, except that a number is appended after the name. In the project options, the default setting of 1 creates one backup file in the Old Versions folder. If you are working with a very complex model, you might decide to specify additional backup versions by changing this setting to a higher number; however, remember that with each additional backup version you are creating additional files (and using additional space) on your hard drive. Setting Old Versions to -1 will cause Inventor to save all backup files.

RESTORING AN OLD VERSION

Occasionally a file may become corrupt, or you may have accidentally saved a design change that you did not intend to. In these cases, you can browse to the Older Versions folder and open the versioned file. Upon doing so you will be presented with three options:

- ◆ Open Old Version (Save Not Allowed)
- ◆ Restore Old Version To Current Version
- ◆ Open Current Version

File-Naming Conventions

The Using Unique File Names setting in the options forces the user to create unique part names for all files in the project, including subfolders. Libraries are excluded in this option. Proper design workflow demands that each unique part have a unique name. When a part is reused, you should ensure that any revision to it is acceptable to all designs in which it is used. If the revised part cannot be used in all of the designs, you should use a new part name because you have now created an additional unique part.

Setting the Using Unique File Names option to Yes will cause Inventor to search the entire project workspace to compare filenames. This can cause issues when a large number of files are present and particularly when those files are organized with a large number of subfolders. If the Using Unique File Names option is set to No, then Inventor will not search to compare filenames.



Real World Scenario

The Using Unique File Names Setting and Slow Search Times

Consider the following scenario: The Using Unique File Names option for your project is set to Yes. You have an engineering directory with 8,500 folders and subfolders containing thousands of files. You have a drawing file that references a part file named 12-865 . i pt, but you've renamed the part file to 22-865 . i pt using Windows Explorer.

Because the internal link in the drawing files is still looking for 12-865 . i pt, when you open the drawing, Inventor will search through all of the 8,500 folders and subfolders looking for the missing file. When it doesn't find the file, it will finally present you with the File Resolution dialog box allowing you to point the drawing file to 22-865 . i pt.

Due to the number of folders and files, this search may take several minutes. By contrast, if you had the exact same scenario but your project file option Using Unique File Names was set to No, Inventor would not search for the missing file. Instead, it would immediately present you with the File Resolution dialog and allow you to point the drawing to 22-865 . i pt right away. For this reason, it is recommended that you set this option to No when large file collections are present. However, even if it is set to No, you should employ a unique file-naming scheme for all of your files.

If you do not have a part numbering scheme already, you should take the time to implement one to make working with your Inventor files easier. Keep in mind that the most effective numbering schemes are often the simplest. Many an engineering department has eventually run into unanticipated limitations when using a numbering scheme that is too specific or when attempting to include too much information in some built-in code. Here are a few suggestions that may help you in determining a numbering scheme that will work well for you:

Sequential project-based: 0910-00001 Here, the first four digits correspond to the project number and the last five are sequenced, starting at 00001. This numbering system works well when parts are not often used across different projects. Common parts might be created under a "common part project" prefix such as 7777.

Generic date-based: 09-0707-01 Here the first six numbers are assigned by using the current date when the part number is created. In this example, the part number was created on July 7, 2009. The last two digits are sequenced starting with 01. This is a highly flexible numbering system that allows 100 part numbers to be created per day. If more than a hundred are needed, then backdating using an unused date can be done. The date itself hold no real significance, other than helping to ensure the unique part numbers.

Sequential product-based: NG-00001 Here the first two letters reflect a specific product line, such as the Next Generation (NG) line, and the last five are sequenced starting at 00001. This numbering system works well when products lines are engineered and maintained separately.

Once you've decided on a part numbering scheme, you will want to come up with a central part number log file or tracking system to be used in assigning numbers to ensure that there are no duplicates. If you have a resource planning system (commonly referred to as a Manufacturing

Resource Planning (MRP) or Enterprise Resource Planning (ERP) system), you likely have the ability to manage part numbers using that software. If not, a simple XLS spreadsheet file can be used to assist with this task.

PART NUMBER LOG SAMPLE

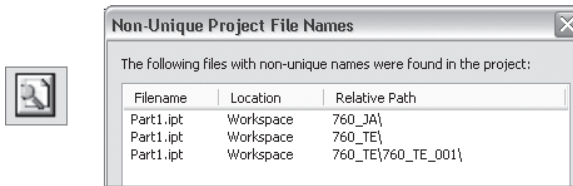
You can find a sample spreadsheet file called `Part_Number_Log.xls` in the Chapter 02 folder of the tutorial files. If you have not downloaded the tutorial files yet, please refer to the steps in the introduction of this book for instructions.

THE PROJECTS DIALOG BOX'S TOOL PANEL

The tool buttons along the right side of the lower pane of the Projects dialog box provide access to tools that allow you add, edit, and reorder project parameter settings and paths; check for duplicate file names; and configure the Content Center libraries used for the active project.

Use the magnifying glass icon located on the lower-right side of the Projects dialog box to check your project paths for duplicate filenames, as shown in Figure 2.10.

FIGURE 2.10
The Non-unique
Project File Names
dialog box



WHY RELATIVE PATHS?

An Inventor assembly file records relative paths when it links a subassembly or single parts to itself. The use of relative paths in assembly files allows the relocation of an assembly and its associated parts and subassemblies to other locations on servers or drives without requiring the resolution of a new location. Relative paths, however, introduce the danger of the assembly locating only the first of two parts that happen to have the same name. For instance, if you've saved files named `Part1` in two different file folders, Inventor might resolve the link to the first one it finds and then stop searching.

To prevent the possibility of the wrong part being loaded in an assembly, it is important for every part located in the search path to have a unique name.

The Projects dialog box supports the configuration of one or more Content Center libraries. The Content Center provides multiple database libraries that can be used in assemblies or by the Design Accelerator (Functional Design System).

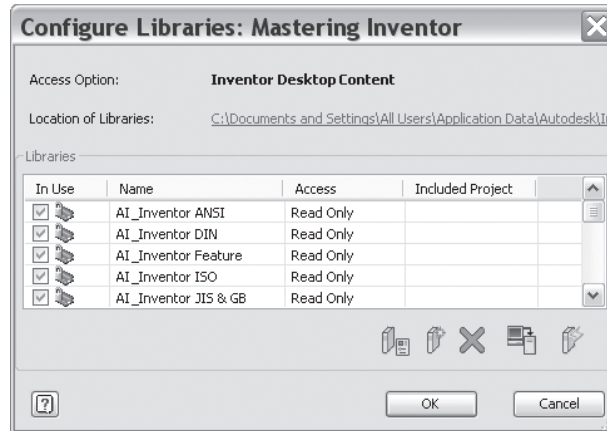
If you elected to install Content Center libraries while installing Inventor, you must configure the Content Center libraries in the project before you can access them. Click the Content Center icon in the lower-right corner of the project-editing dialog box. Then select

the Content Center library or libraries you want to use, and click OK. Figure 2.11 shows the Configure Libraries dialog box.

Select the Content Center libraries you think you'll use. Installing all the Content Center libraries may slow your system down significantly when you are accessing Content Center because Inventor will need to index each library upon initialization.

When you finish editing the project file, click Save, and then make sure your desired project file is active before clicking Done to exit the Projects dialog box.

FIGURE 2.11
Configuring Content Center



Creating Multiuser Projects

Working as a team can increase productivity many times over. In a collaborative design environment, multiple users may be working on a project at the same time. When you create a multiuser project, you have the option to choose Vault (if Vault is installed), shared, or semi-isolated project types. As stated earlier, Vault is similar to a semi-isolated project in how it works. It prevents you from working on the original version of a file located inside Vault. Each user creates a local Vault project file that specifies a personal workspace located on the local drive and includes search paths to one or more master projects.

WHAT IS AUTODESK VAULT AND WHO NEEDS IT?

Autodesk Vault is a file management tool integrated with Inventor. Essentially, Vault allows you to check in and check out files in a collaborative workgroup so users do not accidentally save over one another's work. Not every design department needs to use Vault, and in fact many find that it is not a good solution for their particular needs. However, it is recommended that you investigate Vault to see what it has to offer, even if you already have a data management system in place.

To edit a "Vaulted" file, the user must check the file out of Vault. The process of checking the file out copies the file to the local workspace. When someone checks out a file for editing, the original stored in Vault is flagged as "checked out" to that particular user. Other users can view the checked-out files in read-only mode, but they can't edit it.

The user who checked out the file can edit and save the file in their local workspace without checking the file back into Vault. When they save the file, they will be prompted to choose whether they want to check the file back into Vault. If they choose to check the file into Vault, the file will be saved into Vault and is then available for editing by a different user. Optionally, they may save the file into Vault but keep it checked out to their local workspace, allowing other users to view the updated file without being able to edit it.

PROJECT SHORTCUTS

If you right-click on a project in the Project dialog box, you can choose Delete to remove it from the list. But if you browse to the file location, you'll see that the IPJ file is still there. What is going on?

When you create a new project file or point Inventor to an existing project file, Inventor will create a shortcut to that file. When you choose Delete and remove the project from the list, you are not actually deleting the IPJ file but instead deleting the shortcut. When you choose Browse and locate the IPJ file to add back to the list, the shortcut is re-created.

The path to the shortcuts can be set by accessing the Tools tab of the Ribbon menu and choosing Application Options. In the Application Options dialog box, click the Files tab, and set the projects folder path.

Collaborative design project files are created using the Inventor Project Wizard in much the same manner as a single-user project file is created. The file resolution process within a collaborative project file functions in the same way.


With Vault installed on your server or your own system, you can create and configure a Vault project. If Vault Explorer is not installed on your system, you cannot install or create a Vault project on your system. Before you create your first Vault project, verify that Vault is correctly installed and that you can open and create a new Vault file store using the ADMS console. The new Vault file store must be accessible on your local system from Vault Explorer. If Vault functions correctly, you are now ready to create a Vault project file. As with a single-user project, use the Inventor Project Wizard to name the project, specify the workspace, assign libraries for use with the project, and configure project parameters.

Again, as in other project file types, you will need to edit the default settings in your project file and optionally configure your Content Center for use.

INFORMATION ON IMPLEMENTING AUTODESK VAULT

Once the Vault software is installed on your system you can find the Autodesk Vault Implementation Guide in PDF in the Help folder of the install directory. Or you can search the Internet for "autodesk + vault + implementation + guide." This detailed guide provides you with information on Vault fundamentals and installation as well as information on configuring and maintaining Autodesk Vault for your data management needs.

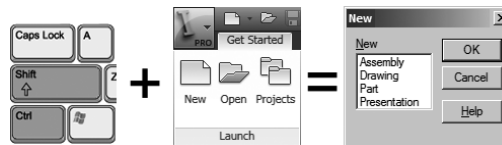
Understanding Inventor Templates

You can create template files in Inventor by opening an Inventor file, making the desired edits to the file, clicking the Inventor button , choosing Save Copy As, and then choosing Save Copy As Template. It is typically recommended that you set the Template directory by configuring your project file using the Folder Options node and setting the Templates path to a network location. Often when creating a template, you might set up the following items in the file before saving:

- ◆ Custom iProperties
- ◆ User parameters
- ◆ Cached styles
- ◆ Specific document settings such as Units (Click the Document Settings button on the Tools tab.)

When creating any template file, it is a good idea to work with a file that is as “clean” and uncluttered with extra styles as possible. The best way to do this is to hold down the Ctrl and Shift keys and click the New button; then select the type of file you want to create from the list, as shown in Figure 2.12.

FIGURE 2.12
Creating a new clean file



Working with Styles, Style Libraries, and Company Standards

Your Inventor files will use a number of *styles* to allow you to maintain consistencies and to save you from needing to define common setups over and over. In part files, the style collections include lighting styles, color styles, and material styles. Color and lighting styles allow you to change the appearance of your model, and material styles allow you to change its physical properties. In drawing files, styles collections include such things as dimension styles, balloon styles, parts list styles and much more. These drawing style collections are organized in *standards*, which are used to manage the standard company styles used within your organization. You can even have multiple standards set up, allowing you to change from one set of styles to another quickly and consistently.

UNDERSTANDING STYLES IN PART AND ASSEMBLY FILES

Although Inventor installs with a number of predefined common styles, you can edit or create new styles to suit your needs. For instance, if you create a lot of parts out of medium-density fiberboard (MDF), you might want to set up a color style to set the appearance of your parts to look like MDF, and you might want to create a material style using the density and other physical specifications of MDF from a supplier data sheet. Doing so allows you to achieve accurate physical properties from your models that can be retrieved in the drawing or stress analysis environments. You could set the MDF material style to use the MDF color style so that any part

that uses the MDF material style automatically uses the MDF color style. You could later apply another color style as an override in cases where MDF is used but the part is then painted.

CREATE A NEW COLOR STYLE

Creating a new color style is very similar to creating a new material. To create a new color style, you would follow these general steps:

1. On the Manage tab in the Styles And Standards panel, click the Styles Editor button to open the Style And Standard Editor dialog box.
2. In the dialog box style list, select a color style that is close in appearance to the one you want to create, and then click the New button.
3. In the New Style Name dialog box, enter the name of the new style, and then click OK.
4. Set the attributes as desired. (For further explanation of each attribute, you can press the F1 key on the keyboard to bring up the help topic.)
5. Click the Save button to save the new style in the current document, and then click Done.

It is important to understand that when you initially create or edit a style, the new or modified style exists only in the current document. If you intend to have these changes available for all newly created files, you would create or edit the style in a template file and then save the changes to the template directory. Sheet-metal part files include the lighting, color, and material styles found in part and assembly files in addition to sheet metal styles that define bend and corner information. You can learn more about sheet-metal styles in Chapter 6.

MATERIAL SPECIFICATIONS

In addition to the information in supplier data sheets, you can find a lot of information for common materials online at www.matweb.com. This is particularly useful when defining a new material for use with Inventor's Stress Analysis tools.

UNDERSTANDING STANDARDS AND STYLES IN DRAWING FILES

In drawing files (IDW or DWG files), the use of styles becomes more extensive, and styles are used in almost every drawing task. Predefined styles control the default appearance of views, dimensions, balloons, parts lists, and so on. These styles are organized under a named standard. You can set up a standard to match your company standard or use multiple standards if you work with multiple companies and need to create drawings using different collections of styles. You can learn more about creating and managing drawing styles in Chapter 12.

MANAGING STYLES AND STYLE LIBRARIES

When you create or edit a style in any file type, the changes are made in the current file only. There are two methods for organizing and controlling the changes to styles to honor your

company's styles and standards when working with Inventor. The first is to create the styles and standards in your Inventor template files and use the Export and Import tools to add new styles to existing files, and the second is to use a *style library*.

A style library is a file that exists outside of the actual Inventor part, assembly, and drawing files, and it contains and controls the definition of individual style types. For instance, if your Inventor project file is configured to use a style library and you created a new part file that did not contain a material and color style for medium density fiberboard (MDF), you would follow these steps to create and manage the new styles:

1. To access the Styles Editor, in the Manage tab find the Styles And Standards panel and click the Styles Editor button.
2. Create the new MDF color and material styles in the part file.
3. Right-click on the new styles (one at a time) and select Save To Style Library.
4. Once the styles are in the style library, you can simply open an existing part file and apply the MDF material and color styles to the part, and the styles are retrieved from the style library file and cached locally in the part file.

By contrast, if your Inventor project file is not configured to use a style library and you created the new MDF color and material styles, you would need to export the styles from one part file and import it into another and/or import the styles into your template part file so that all new files have the new styles.

To create, export, and import the MDF styles without using a style library, follow these steps:

1. To access the Styles Editor, in the Manage tab find the Styles And Standards panel and click the Styles Editor button.
2. Create the new MDF color and material styles in the part file.
3. Right-click on the new styles (one at a time) and select Export.
4. Select the export destination, specify the name of the exported file (the name ends with `.styxml`), and then click the Save button. Repeat for each new style.
5. Open another part file and access the Styles Editor dialog box as described in step 1.
6. Click the Import button at the bottom of the Styles Editor dialog, and browse to and open one of the previously exported files (ending in `.styxml`). Repeat for each new style.

As you can imagine, managing a large number of style changes using the Export and Import options can be tedious. Keep in mind too that by adding all possible styles to each and every file and template file, you are increasing the file size and load times of your models and drawings. Using a style library, though, allows you to include only the styles being used in a particular file.

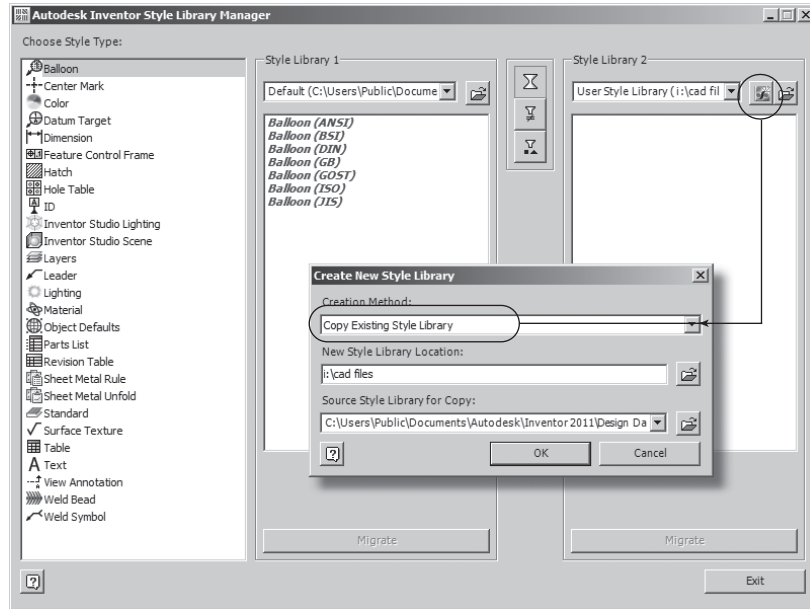
SETTING UP A STYLE LIBRARY

To use a style library, you will edit your project file and set the Style Library option to Yes or Read Only. Typically, the style library is managed by a designated CAD administrator to maintain the integrity of the library. Protecting the style library from unintended changes is done by setting the project file used by all other users to Style Library = Read Only. Configured this way,

the collection of styles in the style library can be accessed by anyone, but edited or new styles cannot be saved to the style library. The CAD administrator will then create a copy of the production project and set the Style Library option to Yes in this project only.

The best way to ensure that you have a clean style library from the start is to create a new, empty style library. You can use the Style Library Manager (choose Start > All Programs > Autodesk > Autodesk Inventor 2012 > Tools) and create a new, empty style library as shown in Figure 2.13.

FIGURE 2.13
Creating a new style library



Once a new style library is created, you can use the >> button to pull existing styles from a source style library into the new library or you can leave the library empty. If you choose to leave it empty, you would typically then edit the project file and set the project to use the new library. Once this is done, you can open existing model and drawing files and use the Save To Style Library option to write the styles found locally in just those files to the new library. A good recommendation is to add color, material, and lighting styles using the Style Library Manager and add the drawing styles, such as dimension styles, from your template once they are created and tested for compliance with your company standard. You can find more information on the Style Library Manager in Chapter 13.

WHERE TO SAVE STYLE LIBRARIES

Inventor installs with a folder called Design Data where the default style library is installed. To locate this folder, you can click the Application Options button on the Tools tab. Click the Files tab in the Application Options dialog box, and look for the Design Data (Styles, etc.) line.

Although you could change this path here, it is recommended that you instead change it by configuring your project file using the Folder Options node and setting the Design Data path to a network location. Be aware that when you do this, Inventor will copy the default Design Data folder (including a style library) from this location. You can then edit or replace this style library as you see fit.

The Bottom Line

Understand how project search paths work. Knowing how Inventor resolves file paths when it opens linked files, such as assembly files and drawings, goes a long way toward helping prevent broken links and repair links that do get broken.

Master It What type of file does Inventor use to point the assembly file to the parts that it contains?

Set up library and Content Center paths. Library and Content Center paths are read-only library configurations set up in the project file.

Master It When you set up a library or Content Center path to a folder that does not exist, what happens?

Create and configure a project file. Project files are a key component of working successfully in Inventor, but for many people, this is a one-time setup. Once the project is created, for the most part you just use it as is.

Master It After creating a project file initially, you want to make one or more changes to the configuration, but you can't seem to do so. What could be the problem?

Determine the best project type for you. Although the Autodesk solution to a multiuser environment is Autodesk Vault, many people may not be able to use Vault. For instance, if you use another CAD application that links files together like Inventor, Vault will likely not know how to manage the internal links for those files.

Master It Because you generally do not work concurrently on the same files as your coworkers, you think it might be best to set up a single-user project for now while you continue to investigate the Vault solution, but you are not sure if that will work. Can single-user projects be used in this manner?

