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CAD Manager Fundamentals

1

CAD Manager Fundamentals

Note on using this guide: In most chapters there are multiple links to topics in the Help. To go directly to the topic, click the link. To return to the CAD Manager guide, click Back.

A resource guide for the CAD Manager

This guide is designed to help the Inventor CAD Manager understand the basic tasks for creating a productive environment within a shared workgroup.

- Installation
- Migration
- Define Application Options and export to your shared server
- Configure the company standard styles using the Styles editor
- Configure the Project (.ipj) file using the Project editor
- Create read-only templates for a shared server
- Configure for optimal performance
- Guidelines for setting up Content Center
- Additional Resources

CAD Manager Best Practices

Before you begin: Set up a shared read-only network location to support templates, and shared design data. This location typically includes your project files, styles library *.xml* files, the application options *.xml* file, and Content Center.

Review the Network and Licensing guides (including the Installation addendum): To access these guides, click the Documentation link on the installation media located on the bottom left-hand corner on each page of the Install Wizard.

Top 5 best practice guidelines:

- Create a good scalable data management structure that supports a single project structure, multiple libraries, and different jobs.
- Create a single project to support the previously described data structure.
- Create a shared location to support templates, design data, and content center.
- Specify the location of the shared Library folder in your project file.



- Use Autodesk Vault to manage all data pertinent to your design, including design files, Microsoft Office documents, images, and other auxiliary data that is worthy of including in the vault. This practice provides a single source for all engineering data.

Consider adopting the following rules:

IMPORTANT While shared network locations in a non-vaulted environment can work for small project teams, to provide secure data management we recommend implementing Autodesk Vault.

- Establish a process for new part creation. Make sure that appropriate data fields are specified and file names conform to a defined naming convention.
- Enforce drawing and tolerance standards.
- Enforce the practice of Constraining a sketch to Origin. Train users to constrain parts in assemblies to **Origin Work Geometry** or **Work Geometry** created from Origin work geometry.

NOTE Origin Work Geometry (and work features created from them) are stable and are not dependent on part faces or features

Additional productivity tips:

- Identify and document modeling workflows that have similar design. Your designers can model similar shapes in a similar manner which creates a stable environment for consuming change.
- Keep a log of your decisions, especially if Autodesk Inventor is new to your company. As you implement the software, you make decisions about how to use certain features or address certain workflows. When another user has the same issue, you can refer back to your notes and stay consistent. This practice also serves as the basis for a Best Practices manual.
- Require regular training to ensure that designers are using similar methods.

Installation and Deployment

The Installation Guides located on the Installer on the Documentation page provide detailed information on how to:

- Install Content Center - Determine whether a local install (Desktop Content) or network install (Vault Server) is best suited for your work environment
- Plan your installation
- Set up Network licenses and create deployments.

To access this content: Start the Installation Wizard from the Autodesk Inventor installation and click:

- Documentation located on the first dialog box displayed by the Install Wizard
- **The Documentation link**, located on the bottom left-hand corner on each page of the Install Wizard.

The Installation Help, accessed from the links on the left side of the Install Wizard also contains instructions for installing and authorizing Autodesk Inventor for both single and networked installations. In addition the Installation Help provides information for installing Desktop Content (Content Center libraries for a standalone environment).

Installation and setting up network licenses requires careful planning and execution. Quick, step-by-step instructions located on the Documentation page explain how to set up a network licensed environment for Autodesk Inventor.

The Licensing and Network Installation guides assume that you are familiar with the terminology and processes required to set up a networked licensed environment successfully. If you are not familiar with network licensing, read the entire Network SAMreport-Lite (Optional) Licensing Guide for further details before you attempt to set up.

IMPORTANT If you upgrade from a previous release of Autodesk Inventor, identify your migration strategy before installing a new release of Autodesk Inventor. If you use Autodesk Vault or Productstream, Content Center and the Styles library, your migration strategy includes four parts: See [Migration](#) on page 17

- Migration of work in progress files.
 - Migration of the server to the new release -For information on the Vault Server, please refer to the Implementation Guide located on the Autodesk Vault Server installer Documentation page.
 - Migration of Content Center libraries.
 - Migration of the Styles library.
-

License timeout, borrowing, and network connectivity

You can set up an options file to subject your network licenses to a specified timeout period. The license timeout feature functionality is not to be confused with the network connectivity requirement (heartbeat).

License timeout functionality

- In the options file, you can set a timeout period for your network licenses. If set, the timeout period is a minimum of 15 minutes, but can be longer.
- If Autodesk Inventor or add-in activity is not detected on the client computer during the timeout period, the license is sent to the network license server so that it is available for other users.
 - If the license is returned to the server, the client attempts to obtain a license from the server when use of Autodesk Inventor resumes.

- If a license is not obtained 30 minutes after the use of Inventor resumes, a message warns that 90 minutes remain before Inventor shuts down. The client continues to search for an available license and notifies if one is obtained. If a license is not obtained, additional warning messages appear on the client to specify how much time remains before Inventor shuts down.
- If a license is not obtained by the end of the 90-minute period, Inventor shuts down. Work can be saved, but other features are unavailable before Inventor closes.

License borrowing

- You can set up license borrowing for your network licenses.
 - Clients can borrow licenses to work with Inventor while disconnected from the network licensing server.
 - In the options file, you define the number of licenses available for borrowing, as well as the maximum borrow period. You can also define which users are eligible to borrow licenses.
 - The user borrows a license by selecting Help ► About Autodesk Inventor ► Product Information ► Borrow License.
 - When the borrow period expires, the borrowed license is automatically released. The user can also elect to return a borrowed license before it expires.

Network connectivity requirement (heartbeat)

- The client computer must be connected to the network so that the license server can monitor the state of the license.
 - To obtain a license, the client must be connected to the network before Inventor starts.
 - If the client becomes disconnected from the network while in possession of a license, the server revokes the license.
 - If the network connection is not re-established after a 30-minute interval, a message warns that 90 minutes remain before Inventor shuts down. Unless the client is reconnected to the network, additional warning messages appear on the client to specify how much time remains before Inventor shuts down.

- If a network connection is not re-established by the end of the 90-minute period, Inventor shuts down. Work can be saved, but other features are unavailable before Inventor exits.

The options file can also be used to set up license borrowing.

Refer to the Autodesk Network Licensing Guide for more details.

Content Center installation and setup

You can work with Content Center as a stand-alone user (libraries are saved in the Desktop Content location on your computer) or as a member of a workgroup (libraries are shared on a Vault server).

If you select the Desktop Content option in the installation, Content Center libraries are installed to the folder specified in the installation configuration. After the installation, the Desktop Content environment is set up and Content Center is ready for use. If you select the Vault Server option in the installation, Content Center libraries are not installed on your computer. You must install Content Center libraries on the server, and log in to the server to use Content Center.

TIP Migrate your legacy user libraries to use them with the new version of Autodesk Inventor.

Before you start using Content Center, set up Autodesk Inventor and Content Center libraries to correspond with your needs:

- Set the Application Options. The Content Center tab in the Application Options dialog box sets the storage location of Content Center libraries and refreshing of standard parts on instancing. The File tab in the Application Options dialog box specifies the default Content Center Files folder where instanced Content Center parts are saved.
- Set Content Center options in the Autodesk Inventor project. Use the Configure Libraries dialog box to configure Content Center libraries. If appropriate, change the Content Center Files folder.

After a successful installation and configuration of Autodesk Inventor Content Center, the following components are available:

- Standard Content Center libraries installed in the Desktop Content folder or on an Autodesk Vault Server and configured in the project.

- One or more user libraries located in the Desktop Content folder or on an Autodesk Vault Server and configured in the project.

Content Center environment for a stand-alone user

If you want to work as a stand-alone user, perform the following steps to set up Content Center:

- Select the Desktop Content option in the Autodesk Inventor installation.
- If your legacy user libraries reside on the server, transfer them to Desktop Content location. Autodesk Inventor can do the transfer automatically on the first startup. Click Yes in the Content Center Library Migration dialog box.
- If appropriate, copy user libraries from an external Desktop Content location to your Desktop Content folder.
- Set up the configuration of Content Center libraries in the project:
 - Transfer user libraries from a server by using Library Transfer guide.
 - Select the In Use box to add libraries in the configuration.
 - Migrate and synchronize legacy user libraries by using the Update Tool.

TIP Legacy and new version of Autodesk Inventor on the same computer:

Use separate folders for old and new Desktop Content libraries. Remember that migration in Update Tool irreversibly changes user libraries. You cannot use migrated Desktop Content libraries in an old version of Autodesk Inventor.

Content Center environment for a design workgroup

You can eliminate the need for each member of a design workgroup to install and synchronize their own Content Center libraries by installing Content Center libraries on a Vault server.

Local settings for workgroup members

After Content Center libraries are set up on the server, members of the workgroups configure their local settings.

We recommend that you designate an administrator to install the server, set up projects and predefine the standard set of libraries to use in a design project. The administrator can create a common project and add shared libraries to

the Content Center configuration, and then share the common project file (.ipj) with the other team members.

To log into the server within Autodesk Inventor, you specify the location (or name) of the remote server in the Log In dialog box.

Storage folder for instanced standard components

When you insert a Content Center library part in an assembly, the library part is copied from the server to the Content Center Files folder. The configuration of the Default Content Center Files folder is saved in Application Options, and the setting can be overridden in the project.


Set the Content Center Files folder depending on how you want to consume changes to the Content Center libraries:


- Specify a network location of the Default Content Center Files folder on each of the local computers for each user to get the most recent library content automatically.
- Specify a location on the local computers for users to decide when to update their libraries. In this scenario, use the Refresh Standard Components command to get the latest content.


Separate servers for Content Center and Vault

The Vault server manages libraries for Content Center and file databases for Autodesk Vault. If needed, you can use two separate servers for Content Center libraries and Autodesk Vault files. Install the server on two computers, and then use one as a Vault server and the other as a Content server:

- To enable the connection to two separate servers, change the Autodesk

Inventor settings on computers of the workgroup members: Click  **► Vault Server ► Connection Options** and select **Use Separate Servers** for Content and Vault in the Connection Options dialog box.

- To log in to the Content Center server, click  **► Vault Server ► Content Center Log In**.

- To view the status of the server, click  **► Vault Server ► Connection Status**.

Comparing Desktop Content and Vault Server

	Desktop Content	Autodesk Vault Server
Installation	Autodesk Inventor installation includes Desktop Content libraries	Autodesk Inventor without libraries installed on client computers, Vault Server and libraries installed on the server
Additional software needed	No	IIS or AWS, SQL Server
User accounts and permissions	No (can be added on the file access level of the operation system, if appropriate)	Yes (managed by Vault Server)
Supports multiple Autodesk Inventor versions on one computer	Yes (with no limitations)	Yes, with limitations (one version of Vault Server, the corresponding and two previous versions of Autodesk Inventor, Autodesk Inventor Client Update installed for legacy Autodesk Inventor versions)
Management of Content Center libraries	In Autodesk Inventor, by using the Configure Libraries dialog	External, by using Vault Server Console
Work with Content Center	No log-in or permissions needed, all Content Center functionality available	Connection and log-in to the server required, editing permissions needed to perform editing tasks
Migration of legacy user libraries	Yes (transfer from ADMS or copying from a legacy Desktop Content folder and migration by Update Tool)	Yes (migration and set up on the Vault Server, then migration by Update Tool)

Working with Autodesk Inventor 2009, 2010, and 2011 libraries on the same server

Autodesk Inventor 2011, the Autodesk server 2011, and the server Client Update installation for Autodesk Inventor 2009 and 2010 enable you to access Autodesk Inventor 2009, Autodesk Inventor 2010 and Autodesk Inventor 2011 libraries on one library server.

To support Side-by-Side installation, the Autodesk server maintains Content Center libraries that were built for different Autodesk Inventor versions. Libraries are saved in the corresponding Partition of the Autodesk server.

Libraries in Autodesk Inventor 2008 partition are used by Autodesk Inventor 2009 with Client Update patch. Libraries in Autodesk Inventor 2010 partition are used by Autodesk Inventor 2011 with Client Update patch. Libraries in Autodesk Inventor 2011 partition are used by Autodesk Inventor 2011.

The default prefix of the library name is determined by the partition where the library is saved. You can copy user libraries from Autodesk Inventor 2009 or 2010 partition into Autodesk Inventor 2011 partition. Copying from Autodesk Inventor 2011 partition to Autodesk Inventor 2009 or 2010 partition is not supported.

Share libraries across design workgroups

Before beginning, designate an administrator who installs and manages Content Center libraries on the Autodesk data management server.

Library configurations for team members are saved in the Autodesk Inventor project and can be edited in Project Editor. Each team member can create a personal project using the project file (.ipj) as a template, or include a common project (.ipj) file that is configured with the shared libraries. This enables each team member to have the same configuration.

NOTE You cannot include a project file in an Autodesk Vault project file. Autodesk Vault does not support included project files.

If you are connecting to a vault database and the Content Center libraries database, the administrator creates the vault project file first and makes it the active project. Then the administrator configures the Content Center libraries in the Configure Libraries dialog box.

Overview

- Install the server and client components for a design team.
- Create a common project file (.ipj) with the desired configuration of Content Center libraries. This project can be used as a template or as an included project.
- In the common project, change the folder where standard components are created, if needed.
- In Autodesk Data Management Server Console, create the user accounts for anyone requiring editing permissions. Communicate user account information and the Login procedure to each member in the team.

Install the server components

Install Content Center libraries on the server. For installation details, please refer to the Autodesk Vault Server Implementation Guide. The guide is installed as a PDF in Program Files ► Autodesk ► ADMS [version] ► Help.

TIP If appropriate, use two separate servers for Content Center libraries and Autodesk Vault files. Install the server on two computers, and then use one as a Vault server, and the other as a Content server.

Use the server console to configure libraries on the server

Before you start using Content Center, set up Content Center libraries to correspond with your needs. Use the server console to create, attach, import, export, detach, and delete Content Center libraries, and to change a library status.

- 1 To run the server console, click Start ► Programs ► Autodesk ► Autodesk Data Management ► Autodesk Data Management Server Console [version].
- 2 In the server console window, expand the root folder and perform the appropriate changes:
 - To create a library, right-click the Libraries entry and choose Create from the menu.
 - To attach a library saved in the default libraries folder, right-click the Libraries entry and choose Attach Library from the menu.
 - To import a library, right-click the Libraries entry and choose Import Library from the menu.
 - To detach a library from the server, right-click the library name and choose Detach from the menu.

Create Content Center Editor user accounts in the server console

Create Content Center Editor user accounts for anyone requiring editing permissions.

- 1 Run the server console: Program Files ► Autodesk ► Autodesk Data Management ► Autodesk Data Management Server Console [version].
- 2 Click Tools ► Administration.
- 3 In the Administration dialog box, click Users.

- 4 In the User Management dialog box, click New User.
- 5 In the New User dialog box, enter the appropriate information for the user.
- 6 Click Roles to assign the user a Content Center role.
- 7 Select Content Center Editor to assign editing permissions.
- 8 Click Vaults to assign the user to a vault.
- 9 If needed, click Groups to assign the user to a group (optional).
- 10 Click OK.
- 11 In the User Management dialog box, and then in the Administration dialog box, click Close.
- 12 Communicate user account information and the Login procedure to each member in the team.


Install the client components

On the local computers, install Autodesk Inventor. Select the Use Vault Server option to skip installing Content Center libraries.


Connect to the server from Autodesk Inventor


- 1 Start Autodesk Inventor.



- 2 Click  ► Vault Server ► Log In.
- 3 Enter the appropriate data to the Log In dialog box:
 - User Name: *Enter the user name.*
 - *Enter the password.*
 - Server: *Enter the name of the computer (for example: MyRemoteServer), the IP address (for example: 141.111.111.111), or the full computer name for your network domain (for example: pc-name.xyz.autodesk.com).*
 - Database: *Enter the name of a Vault database located on the specified server.*
 - Select Automatically login next session to get automatically logged in the next Autodesk Inventor session.

TIP If you use two servers for Content and Vault, change the setting on

computers of the workgroup members: Click  ► Vault Server ► Connection Options and select Use Separate Servers for Content and Vault in the Connection Options dialog box. To log in to the Content Center server,

choose  ► Vault Server ► Content Center Log In. To view the

status of the server, choose  ► Vault Server ► Connection Status.


Create a project file (.ipj) and configure Content Center libraries

Library configurations are saved in the Autodesk Inventor project and edited in Project Editor.

After the common project is configured, copy the project file (.ipj) to a shared location. Request each team member to use the project as a template or as an included project.

TIP You can create several projects with different settings that are specific for your company projects.

Configure Content Center libraries

- 1 Click  ► Manage ► Projects.
- 2 In the Projects dialog box, create a project file or select an existing project file.
- 3 In the Projects dialog box, click the Configure Content Center Libraries command.
- 4 In the Configure Libraries dialog box, configure Content Center libraries. Remove all libraries you do not use and add libraries you want to use. If needed, migrate old libraries.
- 5 Click OK to close the Configure Libraries dialog box.
- 6 Click Save in the Projects dialog box.

NOTE To increase performance, remove all Content Center libraries that are not used by the workgroup. In the Configure libraries dialog box select a library and click Remove Library. If needed, removed libraries can be added back using the Add Library command.

Optionally, change the folder where standard components are created.

When Content Center creates a component, it first searches a cache of previously utilized content located in the Content Center Files location to improve access time to commonly used components. Although it is possible to set different Content Center Files location, we recommend that you use the same Content Center File location for all projects to optimize the performance.

- 1 In the Project dialog box, expand Folder Options.
- 2 Right-click the Content Center Files entry, and enter the new location of the Content Center Files folder.
 - Specify a network location for all users to share the same cache memory of instanced components. The network location ensures that everyone in the shared environment is always accessing the same standard parts and automatically gets the most recent library content.
 - Specify a location on local drives for users to decide when they want to refresh their libraries. In this scenario, a user must click Manage tab ➤ Content Center panel ➤ Refresh to get the latest content.
- 3 Click Save and Done in the Projects dialog box.

Configure local project settings

Connect to the server to change the library configuration in the Autodesk Inventor project.

Use one of the following methods to create a local project:

Create a local project which uses the common project as an included file

NOTE You cannot include a project file in an Autodesk Vault project file. Autodesk Vault does not support included project files.

- 1 In Autodesk Inventor, in the Project dialog box, create a new personal project file.
- 2 Double-click the new project to make it the active project.

- 3 In the Project dialog box, highlight Included files, and then click the Edit selected item.
- 4 Browse to the location of the common project file (.ipj) configured by the administrator.
- 5 Click Open.
- 6 Click Save and Done in the Projects dialog box.

Use the common project as a template for a local project

If you create a personal project file with the shared project as a template, you can save filters and favorites locally.

TIP Select the common project before you click New to use the selected project as a template. The new project has the same library definitions as the common project.

- 1 In Autodesk Inventor, in the Project dialog box, add the common project file (.ipj) created by the Administrator to your projects:
 - 1 Click Browse.
 - 2 In the Choose Project file dialog box, select and open the common project file (.ipj) created by the administrator.
- 2 In the Project dialog box, highlight the common project file.
- 3 Click New.
- 4 Create a new personal project file.
- 5 Click Finish to create the project.
- 6 If appropriate, edit the new project.
- 7 Double-click the personal project to make it the active project.
- 8 Click Save and Done in the Projects dialog box.

Set up the Content Center environment for a stand-alone user

If you work as a standalone user, we recommend you to work in the Desktop Content environment. You do not need to log in to a server or set any user permissions. Libraries are treated as files in the Desktop Content folder on your hard disc. Work with libraries, for example creating, deleting, moving, or copying libraries, is more convenient.

Step 1: Install Autodesk Inventor


Select the Desktop Content option in the Autodesk Inventor installation. Then select Standard Content Center libraries to install. Libraries are installed in the Desktop Content folder on your computer.

Step 2: If appropriate, transfer legacy user libraries on the first startup

When you first run Autodesk Inventor, Content Center looks for legacy user libraries saved on your computer. If a legacy ADM Server is installed on your computer and legacy user libraries are found, Content Center automatically starts transfer of legacy libraries into the Desktop Content location. Click Yes in the Content Center Library Migration dialog box to perform the transfer.

Step 3: Configure Content Center libraries in the project



- 1 Click  Manage ► Projects.
- 2 In the Projects dialog box, click the Configure Content Center Libraries command.
- 3 Use the options in the Configure Libraries dialog box to customize the configuration, and then click OK.
- 4 Click Save and Done in the Projects dialog box.

NOTE The fewer libraries you configure in a project, the better Content Center performs.


Tips:

- Use Library Transfer guide to manually transfer libraries from a Vault server to the Desktop Content location.
- Copy external Desktop Content user libraries to your Desktop Content folder: Locate a library to copy, and copy it as a file to your Desktop Content folder. The library appears in the Libraries list of the Configure Libraries dialog box, and can be added to the configuration.

NOTE The copied library must be unique in the Desktop Content folder. That means it must have a unique File Name and internal library name. The internal library name is created when the library is created, and cannot be changed. If more libraries with the same internal name exist in the Desktop Content folder, they are duplicates even if they have different File Names and Display Names. Content Center configuration does not allow adding libraries duplicate to currently configured libraries.

Step 4: If needed, change the folder where standard components are saved

To change the Content Center Files folder in Autodesk Inventor project:

- 1 Click  ► Manage ► Projects.
- 2 In the Projects dialog box, expand Folder Options.
- 3 Right-click the Content Center Files entry, and select Edit. Then set the new location of the Content Center Files folder.
- 4 Click Save and Done in the Projects dialog box.

If appropriate, change the default folder where standard components are created:

- 1 On the ribbon, click Tools tab ► Options panel ► Application Options.
- 2 On the File tab of the Application Options dialog box, change the default folder for Content Center files.
- 3 Click Apply and OK to close the Application Options dialog box.

NOTE When Content Center creates a component, it first searches a cache of previously utilized content located in the Content Center Files location to improve access time to commonly used components. Although it is possible to set different Content Center Files location, we recommend that you use the same Content Center File location for all projects to optimize the performance.

Migration

Updating your release of Autodesk Inventor often means needing to migrate your existing files to comply with enhancements to the Autodesk Inventor file format. When working with Autodesk Vault and Autodesk Inventor, be

aware of the different approaches to migrating data. Choose the method that best suits your needs.

It is important to distinguish between the four types of data that must be migrated for each release:

- Autodesk Inventor files. **Note:** Opening and saving a file creates a version of the file. If you do not want to create new versions of files you are unlikely to modify, it is not necessary to migrate files. For your work in progress files, we recommend that you use Task Scheduler.
- Content Center libraries
- Style libraries

NOTE For information on the Vault Server, please refer to the Vault Server Implementation Guide located on the Autodesk Vault Server installer Documentation page

Legacy design (.ipt,.idw,.dwg,.iam,.ipn) files: Use Task scheduler

We recommended that you use Task Scheduler to migrate files as needed. For information using Task Scheduler, see the Task Scheduler and Autodesk Inventor Help.

Content Center files (R2008, R2009, R2010):

NOTE The type of design data and the age of the data determine the migration method required to ensure successful migration.

- 1 Access the documentation page from the Installation Wizard on the Autodesk Inventor installation.
- 2 Click Documentation at the bottom left-hand corner of the Install Wizard to access the Documentation page.
- 3 Refer to: Content Center for Side-by-Side Installations of Inventor.

In the Autodesk Inventor Table of Contents Help refer to:

- Autodesk Inventor ► Assemblies ► Content Center ► Migrate or synchronize user libraries
- Autodesk Inventor ► Assemblies ► Content Center ► Administrative tasks on the Autodesk server

Refer to the online help for additional detailed information:

- 1 In Autodesk Inventor, click Help.
- 2 Select the Contents tab, and navigate to the topics:
- 3 Autodesk Inventor ► Collaboration and Visualization ► Share data
► [Task Scheduler application](#) on page 21

IMPORTANT Remember to click **Back** to return to the CAD Manager Guide if you click any of the links on this page.

Migrating Autodesk Inventor Data

Updating your release of Autodesk Inventor often means needing to migrate your existing files to comply with enhancements to the Autodesk Inventor file format. When working with Autodesk Vault and Autodesk Inventor, be aware of the different approaches to migrating data so you can choose the method that best suits your needs.

There are two different methods for migrating non-library files stored in the vault. You can use either Task Scheduler or have Autodesk Inventor migrate the data as the files are opened for editing. There are pros and cons to both methods. Using the Task Scheduler guarantees that all files are up to date, but causes the size of the vault to increase. Using Autodesk Inventor to migrate files as needed causes less change in the vault; however, data may not migrate as well from older releases to newer releases when the files are more than three versions of Autodesk Inventor behind.

To migrate data using Autodesk Inventor

As you open Autodesk Inventor files, Autodesk Inventor automatically migrates the data to the current release. Once the files are migrated, check them back into the vault.

NOTE If you open a design that requires migration and it is not checked out, you are not prompted to check out the files. When you save the design, the files are saved locally but not to the vault. Any subsequent changes you make to your design is not saved in the vault unless you manually check the files out first.

To migrate data using the migration utility

- 1 In Autodesk Vault Explorer, check out the entire folder containing the Autodesk Inventor data. It is not necessary to check out the library files at this time.

- 2 From the Windows Start menu, select Programs ► Autodesk ► Autodesk Inventor [version] ► Tools ► Task Scheduler.
- 3 From the Create Task menu, select Migrate Files.
- 4 In the Migrate Files dialog box, select Add Folder to migrate two or more part (.ipt) files. Be sure to first activate the Migrate from Vault check box if you are migrating files from the Vault. You will be prompted to login to the vault if performing a vault migration.

NOTE

- 5 If you are migrating the files now, click the Submit Now check box. If you are setting up a task to migrate the files at a later time, schedule the start time, date, and so on, for the task.
- 6 Click Options.
 - Select the Total rebuild check box. This step is optional, but recommended.
 - Select the Compact Model History check box.
 - Verify that Skip migrated files is selected.

NOTE Skip migrated files must be selected.

- 7 Click OK in the Migration Options dialog box.
- 8 Click OK in the Migrate Files dialog box.
- 9 Once the files are migrated, check the migrated files into the vault using the Autodesk Vault Add-in for Autodesk Inventor to maintain the file dependencies.

To migrate libraries

When migrating vaulted library files, you must have a single user project configured for access to all library files. It is best to create a project file that contains all library paths from your existing project files. All paths must point to the same location as the original project file and have the exact same Library path name.

If the library files do not contain references to files contained in other libraries you may use the default project.

NOTE It is important that no other users access the files in the vault while files are being migrated.

- 1 Check out a copy of all library files to a local directory, outside the current location referenced by other projects.
- 2 Migrate all the library files using Task Scheduler. From the Windows Start menu, Programs ➤ Autodesk ➤ Autodesk Inventor [version] ➤ Tools ➤ Task Scheduler.
- 3 From the Create Task menu, select Migrate Files.
- 4 In the Migrate Files dialog box, browse to the library project file you created.
- 5 Click Add and then select Add Folder.
- 6 If you are migrating the files now, click the Submit Now check box. If you are setting up a task to migrate the files at a later time, schedule the start time, date, and so on, for the task.
- 7 Click Options.
 - Select the Total rebuild check box. This step is optional, but recommended.
 - Select the Compact Model History check box.
- 8 Click OK in the Migration Options dialog box.
- 9 Click OK in the Migrate Files dialog box.
- 10 Once the files are migrated, check the migrated files into the vault using the Autodesk Vault Add-in for Autodesk Inventor to maintain the file dependencies.
- 11 Get a copy of the library files to the current location referenced by other projects.

NOTE We assume that you updated all your Autodesk Inventor seats to the current version.

Task Scheduler

Task Scheduler is a separate application that performs automated tasks and acts as a batch processor in Autodesk Inventor and other applications.

To open Task Scheduler, click Start ► Programs ► Autodesk ► Autodesk Inventor (version) ► Tools ► Task Scheduler.

Use the Task Scheduler to organize and define one or more time-consuming tasks from different types of programs, and execute the tasks in a specified order at a time that you schedule. Close the Task Scheduler window, and the tasks you scheduled run at the specified time.

Task Scheduler contains predefined task managers for executing common tasks, and a custom task manager for defining your own tasks. A Sequential Task manager is provided to set up multiple predefined and custom tasks. The Migrate Files task is used to migrate files directly from the Vault.

In addition to Custom and Sequential tasks, the predefined tasks include:

- Migrate Files
- Update Design
- Publish DWF Files
- Print Files
- Import and Export Files
- Check In to Vault
- Check Out from Vault
- Get Latest Version from Vault
- Convert IDW to DWG
- Refresh Standard Components in Assemblies

More detailed Help is available in the Task Scheduler.

Increase performance in file migration

Access:



Start ► Programs ► Autodesk ► Autodesk Inventor [version] ► Tools ► Task Scheduler.



In the Task Scheduler main menu, click Create Task ► Migrate Files.

TIPS

- In the Migrate Files dialog box, click Options tab ➤ Total Rebuild. Although this method takes longer than selecting an entire project at a time for migration, following these steps ultimately increases Autodesk Inventor performance.
- Always make a backup.
- Migrate all part (.ipt) files first, then all subassemblies, and finally the main assembly.

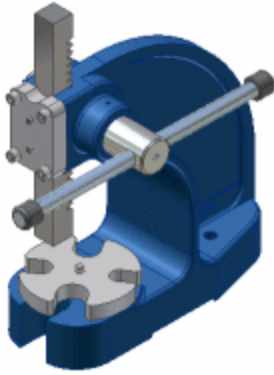
The following method reduces the time to open an assembly model, since Autodesk Inventor does not need to migrate files before opening them. Although this method may slow down the migration, it ultimately increases Autodesk Inventor performance.

NOTE This method is a faster than selecting an entire project at a time for migration within Autodesk Inventor

To ensure the full benefit of Memory Saving Mode, migrate all your Autodesk Inventor R10 and earlier files. To facilitate this task, use Task Scheduler. Access Task Scheduler: Programs ➤ Autodesk ➤ Autodesk Inventor [release] ➤ Tools ➤ Task Scheduler.

- 1 In the Migrate Files dialog box, click Add. Select Add Folder to migrate 2 or more part (.ipt) files, and then select From Local or From Vault.
- 2 Browse to the appropriate folder and click OK.
- 3 Click the down-arrow in the File name column, and select .ipt to display only part files.
- 4 Once completed, repeat the previous steps for all the subassemblies (not the main assembly).
- 5 Repeat the steps for the main assembly.

- 6 Finally, migrate the drawing (.idw) and the presentation (.ipn) files.



NOTE This method is more efficient than selecting an entire project at a time for migration. Using this method, Autodesk Inventor loads each part and subassembly into memory on a one by one basis before freeing up the memory for the item.

Migrating just the main assembly, Autodesk Inventor loads every part and subassembly separately into memory including reoccurring parts and patterned assemblies. This technique can result in "out of memory" and stability issues with large assemblies since Windows on a 32-bit operating system can only allocate a maximum of 2 GB of memory to each CPU process (this 2-GB limit may be spread over RAM or the pagefile). With large assemblies containing Component Patterns, it can start to be a major issue.

Migration of Legacy User-Created Content Center Libraries

Legacy user libraries that contain user-published content or standard content modifications must be migrated to make them usable in Content Center. If your legacy libraries reside on a server, you must first transform the libraries to the new server database schema. We recommend you to transform libraries by the migration process that is started automatically after installation. After you transform legacy libraries on the server, copy them to the server partition for the new Autodesk Inventor version. Then migrate and synchronize libraries by using the Update Tool.

Configure Libraries dialog box lists libraries available in the Desktop Content location or in the corresponding partition of the Vault server. Out-of-date user libraries are marked with the following icon:



Migration needed marks a library that contains data from a previous version of Autodesk Inventor. The library cannot be used in Content Center until it is migrated.



Synchronization needed marks a library that contains data families with a link to parent families and some of the parent families were changed.

The library can be used in Content Center but instanced members of parent families and copied families may differ.

Update Tool

Update Tool is a wizard that migrates the content of legacy libraries into the current Autodesk Inventor version and synchronizes the copied families with link and their parents.

Migration process transfers family templates and family data to a new Autodesk Inventor format. All legacy user libraries must be migrated before they can be used in a new version of Autodesk Inventor.

A legacy user library is marked as Migration Needed, when it is added into the configuration in a newer Autodesk Inventor version.

Synchronization process propagates edits from a parent family to the copied (child) family with link. Data overridden on the family copy are preserved, but all other changes are applied. If you changed the template file, your template file is overwritten on family update.

A family with a link becomes out-of-date if the parent family changes (that means if a new version of the parent family is installed or if the parent family is edited). The parent family must be available in the Content Center configuration to keep synchronization. If a parent family is not available, the corresponding copied family is marked as up-to-date. After you add the library with the edited parent family to the configuration, the copied family is recognized as out-of-date, and its library is marked as Synchronization Needed.

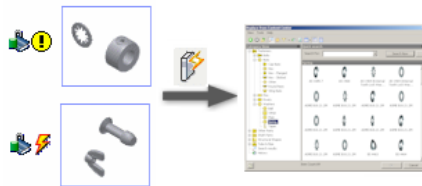
Failures in the synchronization process

The automatic data migration may fail for some families in the library. Then you must fix the families manually. Specifically, the following data or changes may require user interaction:

- User-published families (parts) in categories requiring authored content must be first updated (and re-authored in case that the category definition was changed).
- Families that were copied from the legacy Standard Content Library and members that were added that represent user sizes.
- New category parameters that require column mapping.
- Removed Category parameters.
- Changes in part templates.
- Changes in the family name.
- Edits in family table.

- New Families.
- Part templates of the user-published parts.

Migrate and Synchronize libraries by Update Tool






All user libraries created in Autodesk Inventor Content Center must be migrated before they can be used in a newer version of Autodesk Inventor. Library synchronization is optional.

NOTE If you use the SQL Express server, the limit for size of each library is 4 GB. If a library exceeds the limit during the migration, the migration of the library prematurely stops, and the library cannot be used in new Inventor version. To avoid this problem, split large libraries or install a full version of SQL.

TIP If you store libraries on a server, the installation imports your legacy libraries to server partition corresponding to the legacy Inventor version. To use legacy libraries in the new version of Autodesk Inventor, you must copy them to the corresponding partition. See the [Manage Content Center libraries by using the server console](#) on page 167 page for more information.

TIP Create a backup copy of your legacy libraries before you process them in the Update Tool.

- 1 Click  ► Manage ► Projects.
- 2 In the Projects dialog box, click Configure Content Center Libraries. 
- 3 In the Configure Libraries dialog box, click Update Tool. 

TIP Before you run the Update Tool, select the In Use option for all user libraries you want to migrate or synchronize.

- 4 In the Update Tool guide, review the introductory page, and click Next.

- 5 Select libraries to migrate and click Next.

The Libraries to Migrate list displays legacy libraries that need to be migrated. If you clear the selection for a library, the library will not be available for use in your version of Autodesk Inventor.

- 6 (Desktop Content libraries only) Review the migration report. After the migration ends, click Next.

- 7 Select libraries to synchronize and click Next.

The Libraries to Synchronize list displays all libraries that include out-of-date copied families with link to parents. Parent families must be available in the current library configuration to perform the synchronization.

NOTE Synchronization is optional. You can use a library even if it is not synchronized.

TIP If appropriate, suppress links for families to defer their synchronization. Right-click the families in the Content Center Editor dialog box, and click Suppress Link.

- 8 To save your customized family templates for updated families, specify the backup folder. To skip the backup of family templates, select the Do not Create Backup of Customized Family Part Templates option. To overwrite existing templates in the selected backup folder, select Overwrite Existing Files. The options are not available if no library is synchronized.

TIP User-defined family templates are replaced with templates from the new Autodesk Inventor library for synchronized families. To restore your templates, create their backup copies, and then manually replace the standard templates with your modified templates saved in the backup folder.

To save a log file, select Create Log File, and specify the folder to save the log file.

Click Next to start the process. The Update Tool process may take long, especially if a large amount of data is processed.

- 9 After the process ends, review the process notes and results.

All problems are reported in the dialog box and in the log file. Content that was not migrated successfully cannot be used, and must be fixed manually.

- 10 If you synchronized a library, refresh all instanced members of copied families that were edited on synchronization. Open assemblies with the corresponding standard parts and click the Refresh Standard Components command. Or use Task Scheduler to refresh standard components in more assemblies in one process.

Manual Fixing of Failed Families

Details for fixing a family are provided in a family tooltip in Content Center Editor. Two icons represent two different problems:



New parameter mapping or re-authoring of the part template is needed. The family can be used but may fail in applications.

Open the Family Properties dialog box and map categories. You may need to modify the family template to provide a source for the new mapping to category parameters. Use the Open from Content Center command to open the template file, and modify it. Then replace the family template by using Content Center Editor.

Perform the authoring process for the family template, if needed.




Values are missing in the family table. The family is invalid and cannot be used. Edit the family table and add the missing data to correct the family.

Fixing families that need Authoring

Old legacy content that requires authoring must be re-authored and moved back to the appropriate category to be available in applications, for example in Design Accelerator or Tube and Pipe.



- 1 Click  ► Open ► Open from Content Center.
- 2 On the ribbon, click Manage tab ► Author panel ► Component.
- 3 Author the part for the appropriate category.
- 4 To get the authored part back as a template file for a family:
 - On the ribbon, click Manage tab ► Content Center panel ► Editor.
 - Right-click the appropriate Family, and select Replace part template.
 - Select the new template.
 - Click OK to apply the changes.




NOTE The Authoring command recognizes the category where the part should go and the category (kind of part) is automatically set.

Quick Reference

Update Tool

Use the Update Tool guide to migrate and synchronize user libraries.

Access:

Click  ► Manage ► Projects. In the Projects dialog box, click Configure Content Center Libraries.  In the Configure Libraries dialog box, click Update Tool. 

Step 1: Welcome Page

Review the introduction and click Next.

Step 2: Libraries to Migrate

Select libraries to migrate and click Next.

The Libraries to Migrate list displays all legacy libraries that need to be migrated. If you clear the selection for a library, the library will not be available for use in your version of Autodesk Inventor.

Step 3: Migration Report

For Desktop Content libraries only.

Review information about library migration and click Next.

The report lists the migrated libraries and informs you about successful migration or about migration problems.

Step 4: Libraries to Synchronize

Select libraries to synchronize and click Next.

The Libraries to Synchronize list displays all libraries that include out-of-date copied families with link to parents. Parent families must be available in the current library configuration to perform the synchronization. Synchronization is not mandatory.

Step 5: Library Analysis

Set the options and click Next to start the process.

Select location for family templates backup If appropriate, change the location of the backup folder.

Do not create backup of customized family part templates Select to discard your customized family templates. If not selected, customized legacy family templates are saved in the backup folder, and you can replace the standard templates with your customized templates.

Overwrite existing files Select to replace the existing files in the backup folder with the new files.

Create Log file Select to create a log file. The log file is saved in the specified folder.

Step 6: Update Finished

Review the process report.

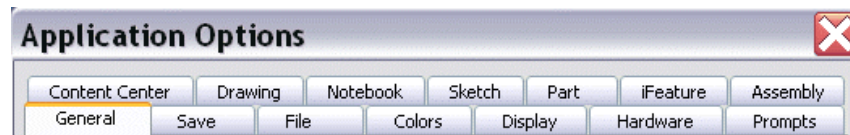
Done lists all actions that are complete.

Final fix required lists families that require additional table fix.

Failures lists families that did not update and why the family failed to update.

Define Application Options and export to your shared server

Consider whether the following Application Options settings are useful for your company.



Of Note:

Files tab: Specify the network locations of the shared design files.

NOTE Ensure that users have network permissions to access this location.

Hardware tab: A certified Direct 3D graphics card using the correct driver offers the best graphics performance.

NOTE For additional information, visit the Autodesk Inventor Hardware web site: <http://www.autodesk.com/inventor-graphic-cards>

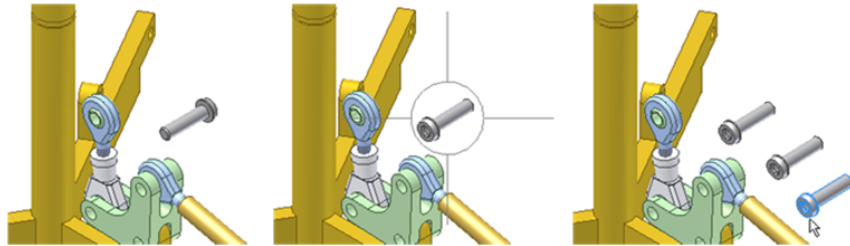
Consider the recommended settings:

General tab: For better viewing, increase the Annotation scale (1.2 to 1.5) to increase the size of sketch dimensions and constraint icons.

Under Selection, disable Enable Prehighlight if users will be dealing with large assemblies or complex parts. Also, adjust Select Other Delay to off.

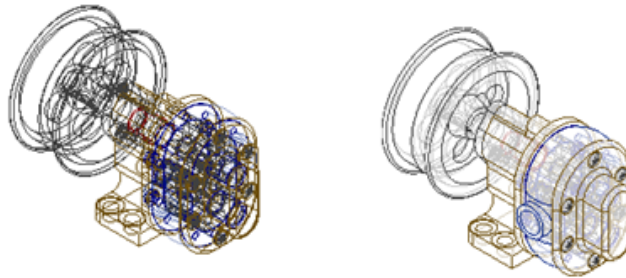
Assembly tab:

Use last occurrence orientation for component placement = On. Enabling this control increases productivity. For example, if you place an object and then rotate or constrain it, all new instances are placed in the original orientation. This following sequence shows the result of this setting turned ON, rotating the first instance, then placing more copies.



Zoom target for place component with iMate = None or All (default is to zoom to placed component). Change the default setting to None or All so that users are not forced to zoom out every time they place a component.

Display tab: Under Wireframe display mode ► Active, turn ON Dim hidden edges.



Colors tab:

- Set the background color if you want to enforce consistency. Alternately, it can be useful to communicate to your designers to set the background color and line contrast to a color that is easy on their eyes.

Sketch tab:

Set the controls:

- Apply driven dimension = On to save time and avoid unnecessary mouse clicks.

- Edit dimension when created = On to automatically activate the edit box once the dimension is placed.
- Autoproject part origin on sketch create = On to automatically create a point on which you can anchor a sketch to.
- Display ► Grid lines = Off to reduce clutter. Tell the designers they can turn on the control when they need grid lines for alignment purposes (such as in *.idw* files).

iFeatures tab:

- Enable the Use Key 1 as Browser Name column option. This will cause any iFeature that has Key 1 set to display their values in the Browser for easier recognition.

For additional tips on setting Application Options, see: [Increase performance and capacity](#) on page 32 Click the Procedure tab, and click: **Application Options tips**.

Save your customized Application Options settings: When you finish configuring the Application Options, use Export to save the current Application Options settings in an *.xml* file.

You can deliver the *.xml* file created using the Export command in the Application Options dialog box during deployment using the 'I would like to Import custom settings' option on the installer. This option is available via the Deployment Wizard on the installation media.

NOTE The default location of the file is specified in *Program Files\Autodesk\Inventor [version]\Preferences*. Define a different location on the File Tab ► Design data.

Increase performance and capacity

Performance

Performance refers to how long it takes Autodesk Inventor to complete an operation on your computer, including loading files.

There are many reasons for performance issues on a CAD system. You can resolve some of them by upgrading to a faster computer or adding more RAM. Keep the CAD system “tuned up” by performing regular maintenance, such as defragmenting the hard drive, also resolves performance issues.

This section covers hardware requirements, best practices, and techniques to use when you have reached a performance impasse. The following information is a guide to areas you can fine-tune to have a positive impact on performance. You can employ only a few of the tips provided to improve performance, or you can employ all of them as your assemblies approach a million parts!

There are options you can configure in Autodesk Inventor that strictly go to improving performance. For example, if you open a drawing file with Defer Updates enabled, you increase performance, but you do not change what Autodesk Inventor loads into memory.

Capacity

Capacity refers to how much memory is required to do an operation combined with how much memory is being used by your computer at any single moment.

Note on 32-bit operating systems: Microsoft Windows has a limit of 4 GB of memory for every process or application such as Autodesk Inventor. By default, Windows reserves 2 GB of that space for the operating system, thus leaving 2 GB for tasks performed by applications.

If you routinely work with large data sets or can resolve performance issues by adding more RAM, consider upgrading to a 64-bit computer which can access terabytes of RAM.

2 GB of memory is not enough for users who commonly work on extremely large assemblies and consume the full 2 GB of virtual memory space. It is important to keep this concept in mind when working with large files, and to determine whether the 3 GB switch feature is an option your computer may benefit from. By changing a few system files, you change the default, so that 1 GB of memory is used by the operating system, and 3 GB are available to applications.

There are also options that can help with both performance and capacity. In an assembly file, after achieving the appropriate result using the assembly selection filters, create a design view representation. When you open a model with a design view representation displaying only the components needed for the immediate design session, you save on both capacity and performance. The graphic data of the components set to invisible are not loaded into memory (thus not sacrificing capacity), and the model opens more quickly.

Click the Procedure tab, and then click Autodesk Autodesk Inventor 3 GB feature (Applies to 32-bit operating systems only) to learn how to set up the 3 GB feature.

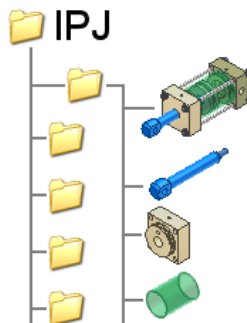
The following table compares the memory capacity of a 32-bit system with a 64-bit system.

General Memory Limits	32-Bit	64-Bit
Total virtual address space (based on a single process)	4 GB	16 Terabytes
Virtual address space per 64-bit process	N/A	8 Terabytes
Paged pool	470 MB	128 GB
Non-paged pool	256 MB	128 GB
System Page Table Entry (PTE)	660 MB to 900 MB	128 GB

Here are some other general considerations:

Defragment the hard drive on a regular basis by scheduling a Windows Task. If Autodesk Inventor files are fragmented, they take longer to open. Run Disk Check before you start.

Plan your folder structure and assembly hierarchy before you begin. Create subassemblies to improve performance over “flat” structures. We recommend you place all subassembly components or related files in the same folder. This does not mean you should put all your files in one large folder. Autodesk Inventor searches all files in a folder. Create as many folders as required. The fewer files in a folder, the faster required files are found. If possible, use a single project file for simplicity and a local hard drive for edits.



Use sketch blocks in a 2D part skeleton sketch to create a schematic layout of your mechanism. Create flexible, nested blocks and apply sketch constraints

to define subassembly kinematics. Derive sketch blocks into component files and create other features to develop your 3D models. The components remain associated to their corresponding blocks and update to reflect any changes in block shape.

Create a multi-body part to define an assembly. Use Make Components to convert the individual bodies into parts and create an assembly with all components grounded.

Increase memory capacity

If you experience memory consumption issues when opening Autodesk Inventor models, refer to the following suggestions.

General

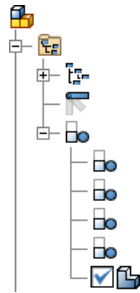
- Keep the modeling Level of Detail of purchased or library components to what is needed for accurate design (such as space envelopes, hole sizes, and locations). Adding unnecessary detail (such as textures, threads, coil features, fillets) can affect performance and capacity.
- Switch to the Classic interface style.
- Close all applications that you do not need opened when working with large assembly files to reduce page swapping with the hard drive.
- Unload any unnecessary add-ins before opening an assembly. Check the Add-In Manager to find out what is loaded on startup to determine if any can be unloaded. For example, do not load Routed systems:Cable and Harness on startup if it is not required.
- Consider not placing hardware parts at all, or only place one instead of many. Quantity overrides can be performed on the Bill of Materials and Parts List to accurately capture the required number of fasteners and other hardware in a design.
- Make it a practice to purge all unused style definitions. Each time a material or color change occurs, the file caches the style definition. If the file is referenced many times in an assembly, the unused definitions can have an impact on memory. An external style library stores material, light, and color style definitions, so maintaining more than one local, cached definition is not necessary. To purge unused style definitions, select Manage tab ► Styles and Standards panel ► Purge.
- Use Selection commands to hide or suppress sets of components based on such factors as size or on internal components that are not seen. Note that BOM reports remain accurate regardless of component suppression.

Parts

- Suppress large feature patterns on parts. Consider using a bitmap texture in place of large feature patterns.
- Reduce unnecessary details in parts. For example, do not model physical threads, fillets, and gear teeth if the detail is not required for manufacturing.
- Minimize the usage of Move Bodies in a multi-body part. Use click to add to group moves into one feature whenever possible.

Assemblies

- Use only enough assembly constraints to achieve the required component position or motion.
- Avoid redundancy. Use the Application Option **Enable constraint redundancy analysis** to check for redundant constraints. Turn off the option after completing the analysis.
- Minimize the use of assembly features.
- Use a common constraint reference if possible. Constraining all components to a common component or geometry improves performance and reduces complexity. For example, use the Origin work features to constrain components in an assembly whenever possible.
- Use a common origin for static assemblies using skeletal modeling.
- Constrain symmetrical assemblies to mid-planes or center axes.
- Locate and fix or suppress any constraint errors. Use the Design Doctor to isolate components.
- Use iMates to reduce overhead and enforce consistency.
- Use component patterns whenever possible.
- Use Level of Detail Substitute representations to replace the entire assembly with a single part file in the same assembly file. The substitute part can be any part file on disk. A substitute surface composite part created using the Shrinkwrap or Derived Component command significantly reduces the memory requirements and file size of an assembly.



- As components are added to an assembly, suppress the components that are not necessary for constraining or locating new components.
- Whenever possible, turn off the visibility of Coils and Springs in assembly files.
- Use Design View representations of large assemblies to display only the components needed for the current tasks. This improves graphics performance.
- Create Design View representations which set specific colors for sub-systems to make them easy to distinguish and manage.
- Use Level of Detail representations to suppress components that are unnecessary for the current task. This reduces the amount of memory in use. Consider creating an LOD from the bottom-up. For example, open an assembly with “All Parts suppressed” and then unsuppress the parts and components you must see or work with.
- Turn off the Contact Solver after performing a contact analysis.
- Turn off Adaptive. Use the Flexible option if the component must adapt to a new position.
- Consider using Grip Snap to position components accurately, and then ground them.

Drawings

- Keep the size of drawing files as small as possible by minimizing the numbers of views on a sheet.
For example, have one base view of the main assembly, and no more than 4 other projected, detailed, or section views.
- Minimize the number of sheets in a drawing file.

- Before inserting a .bmp into a title block, make sure the .bmp is saved in the smallest file size permitted by your company's standards. In Microsoft Paint, the default file type is 24 bit-Bitmap. Reducing the file type to 16 Color or Monochrome type Bitmap, will increase capacity.
- In the Drawing tab in Document Settings, set Always in Bitmap Resolution drop-down menu with a low bitmap resolution to help reduce memory consumption when working with large or complex models of shaded views.
- In the Drawing tab in Application Options, enable **Memory Saving Mode**. Memory Saving Mode instructs Autodesk Inventor to be more conservative with memory both before and during view computation at the expense of performance. It conserves memory by changing the way components are loaded and unloaded.
Tip: To ensure the full benefit of Memory Saving Mode, you must migrate all your Autodesk Inventor R10 and earlier files. To facilitate this task, use Task Scheduler. Access Task Scheduler: Programs ► Autodesk ► Autodesk Inventor [release] ► Tools ► Task Scheduler.

NOTE This option may increase capacity and will have a negative impact on how long it takes Autodesk Inventor to compute data.

Use assembly selection filters to improve performance

Here are some tips for using the Assembly Selection Filters to improve performance in large assemblies and to control what is being loaded into memory:

- Activate the Visible Filter to restrict the tools to only select from visible components.
- Set the selection priority to the appropriate object type to include in the selection set. It is generally useful to set the selection priority to Part priority to ensure all components are considered when creating a selection set.
- Use Visibility to turn off the visibility of groups of components. It speeds up subsequent loading of assemblies or drawing view creation. Upon achieving the appropriate result, create a design view representation to allow fast access to the same results in the future.
- Use the Assembly Selection Filters to create design view representations for drawing files. For example, when you place a view using a design view that was captured by the All in Camera tool, only objects in the Camera's view plane are computed, thus increasing performance and increasing

capacity. This applies to design views created when working with any of the Component Selection commands.

Autodesk Inventor 3 GB feature (Applies to 32-bit operating systems only)

Background

Microsoft Windows reserves the upper half of the 4 GB virtual address space of any process for the operating system, and leaves the remaining 2 GB for the application process, including the space for the code pages, the stack, and all dynamically allocated memory.

Thus, Microsoft Windows operating systems support no more than 2 GB of memory usage for one application. The 3 GB feature divides the memory space differently, providing 3 GB for the application and 1 GB for the operating system. This is not a network server limit, but is relevant to users working with large, memory-intensive data sets.

NOTE When using the 3GB switch, it is recommended that the computer has at least 3 GB of physical RAM, but 4 GB of RAM is preferred. This will considerably reduce the amount of paging done by the operating system. It is also required that the size of the page file should be 4GB or more.

To set the page file size, right-click My Computer ➤ Properties ➤ Advanced Tab ➤ Performance Settings ➤ Advanced Tab ➤ click Change in the virtual memory section. Enter 4096 in Initial size box as well as in Maximum size box and click on Set. Click OK.

To enable the feature

You must change your boot.ini file (which is typically located at C:\boot.ini) to have the 3 GB option, as shown below. It is recommended that you copy the original line, and then modify the copied line to add the option and the identifying string. This sets it up so you can choose at start time whether to use the option or not.

NOTE The default is to use whichever line is first.

The copied line is show in green (last line), with the modified portions of the line shown in red.

[boot loader]

timeout=30

default=multi(0)disk(0)disk(0)partition(2)\WINDOWS

[operating systems]

```
multi(0)disk(0)rdisk(0)partition(2)\WINDOWS="Microsoft Windows XP Professional" /fastdetect
```

```
multi(0)disk(0)rdisk(0)partition(2)\WINDOWS="Microsoft Windows XP Professional 3GB" /3GB /fastdetect
```

To ensure this is working

To insure that Autodesk Inventor is actually allocating memory in the 3 GB range, start Autodesk Inventor. Click Help ► About Autodesk Inventor. The first line of text under the splash screen image should say: "Autodesk Inventor [release number] - 3GB" If the - 3GB is shown, Autodesk Inventor is successfully using the feature.

Limitations

Each executable must be specially built to enable the larger address space. Only the Autodesk Inventor.exe has been enabled. Other executables shipped with Autodesk Inventor have not been enabled. Most likely, the applications provided by other software vendors have not been enabled to use this feature.

The feature only works on Windows XP. Furthermore, the Windows XP SP1 service pack disabled it. To re-enable it for Windows XP SP1, there is a hot fix from Microsoft that must be applied. To apply that hot fix (q328269_WXP_SP2_x86_ENU.exe), follow the instructions in the hotfix.txt file in the associated zip file. It is not possible to enable the feature on Windows NT, Windows 2000, or Windows 9x.

Application Options tips

You may find that by implementing LOD's, Shrinkwrap or Derived substitutes, and using some of the other techniques described previously, you eliminate performance issues. If not, the next section offers suggestions for adjusting the Application Options to consume the least amount of resources.

Access:

Ribbon: Tools tab ► Options panel ► Application Options



The following settings on the tabs listed below can affect performance and/or capacity:

General

Show command prompting = Off

Enable Optimized Selection = On

Enable Prehighlight = Off

Locate tolerance control: Lower values can increase performance.

Undo: Make sure there is enough free space on the specified partition to accommodate the size of the undo file. Use 512 MB to 1 GB for large assemblies.

File

Projects Folder: Working from your local drive is the preferred method. Avoid working on Autodesk Inventor files across a network. When working on a collaborative project, use Vault.

Colors

Background = 1 Color

Display

Appearance Settings

Shadow options can effect performance for hardware supporting feature level 3.1 and lower. To find out which feature level your graphics card supports:

- 1 On the Tools tab, in the Options panel, click Application Options.
- 2 Select the Hardware tab, and click Diagnostics.
- 3 In the displayed dialog box, under the AIRViz Device Manager, you find the information about the Feature Level supported by your hardware.

Turning off shadow effects will improve performance for lower feature levels. It is recommended that you experiment with one level at a time, in the following order, to find the right level of shadow usage.

Ambient Shadows = Off

Onbect Shadows = Off

Ground Shadows = Off

Other options for improving graphics response are:

- Reducing the number of visible components in large assemblies.

- Use simplified representations for components in large assemblies.

View transition time = 0 Inventor manages view transitions automatically, however, you can override the automatic setting by reducing view transition time.

Minimum frame rate = 10 This option displays less graphical information during Pan, Zoom, and Rotate. This is automatically managed, however, the setting is there for you to manually override the automatic feature.

Hardware

Use Direct3D graphics hardware

Be aware that memory consumption varies with different graphic cards. Review recommended video cards and other hardware components on Autodesk's Web site.

Use the default settings on this tab. If you experience stability issues, work your way down the list of available options on this page.

Drawing

To improve performance and capacity:

- Cancel the selection of Display line weights and Retrieve all model dimensions on view placement.
- Enable Memory Saving Mode. This option may allow you to create and edit drawing views of very large and/or highly complex models that would otherwise experience capacity limit issues. This option is also available in Document Settings and can be overridden on a per document basis within a drawing file. Memory Saving Mode

NOTE

- increases capacity but will have a negative impact on performance.
- Drawing view creation and modification operations cannot be undone or reverted while the Memory Saving Mode option is enabled. The Undo/Redo commands in the application will be disabled as a result of this.

-
- Enable one of the Show Preview options, Partial or Bounding Box options to reduce memory consumption. The bounding box option displays the least amount of graphics, and gives the best performance.

- Enable Show Preview options, Bounding Box options to reduce memory consumption. The bounding box option displays the least amount of graphics, and gives the best performance.
- Select Section View Preview as Uncut to preview a section preview as uncut.

Sketch

Leave the Autoproject edges for sketch creation and edit option not selected to improve performance and not sacrifice capacity when not needed

Notebook

Leave Note Icons not selected to increase capacity.

Configure the company standard styles using the Styles Editor

The Styles and Standards Editor offers you a central source for managing styles. Styles control the default behavior of how this information is displayed.

Use the Styles and Standards Editor to set the active standard for a drawing document and to set material and lighting styles for part and assembly documents. For example, define the behavior for balloons, dimensions, layers, line entities, symbols, tables, revision blocks, and so on.

After customizing the Styles Editor to conform to the needs of your company, remove any styles that are not used.

IMPORTANT Put the style library on a network as a read-only file so the whole project team can share the same formatting standards, materials, and so on. When changes are made to a style that all users must consume, save the changes to the Style Library (for example, changing the precision of a dimension style). Changes to templates (potentially multiple templates) are not necessarily needed. Once saved, document styles are updated directly from the library

Configure the project file to point to the styles library on the network. In the Project Editor, set the properties to read-only to prevent designers from changing the styles and standards of the company.

For detailed information on configuring the Styles Editor in the Help:

- 1 In Autodesk Inventor, click Help.

- 2 Select the Contents tab, and navigate to:
- 3 Autodesk Inventor ► Work Environment ► Styles and Styles Libraries:
[Style libraries](#) on page 44

Some workflow tips:

- Create the Text and Layer styles first.
- Modify the remaining object styles as needed
- Customize the Object Default Style to meet the needs of your company.
- Set your company standard to use your Object Default Style

IMPORTANT Remember to click **Back** to return to the CAD Manager Guide if you click any of the links on this page.

Style libraries

Style Libraries are a common source of styles for projects. Style Libraries are shared between users in a manner like custom component libraries.

A style library contains the definition of individual style types. When you apply a style to an object in a document, the attributes of the style are retrieved from the style library.

By default, all styles in the style library associated with the active drafting standard are available for formatting objects in documents. Usually the style library is managed by the CAD Administrator, so that the integrity of the library is maintained across the design project.

You can use a single global library so that all designers use the same styles or you can specify a library for each project. Project style libraries are often a variation of the default library.

A style library can be specified in the project options. For more information, see the Project Editor reference.

How can I share styles across documents?

Styles are shared (or copied) among documents in two different ways:

- The styles can be exported using the Style and Standard Editor Import/Export functionality.

TIP To export a style, open a document, select a style (or styles) in the Style and Standard Editor dialog box, right-click, and select Export from the pop-up context menu. To import a style, click Import at the bottom of the Style and Standard Editor dialog box, and select the style to import.

- The style can be saved to a Style Library, where it can be reused by other documents.

What advantages do style libraries offer?

Style Libraries create a central location of styles for simple administration.

They offer users a central source for styles. Put a style library on a network so the whole project team can share the same formatting standards, materials, and so on. When changes are made to a style that all users must consume (for example, changing the precision of a dimension style), all that is needed is to save the changes to the Style Library. Changes to templates (potentially multiple templates) are not necessarily needed. Once saved, document styles are updated directly from the library.

Style Libraries are a common style source for all documents.

With Style Libraries, different document types read styles from the same source. This feature benefits parts, sheet metal parts, assemblies, and presentation files, which all use the same color and lighting styles from the library. It also means that materials can be shared between parts, sheet metal parts, and weldments.

Style Libraries can aid with large assembly performance.

Without a Style Library, each required style definition must be stored in a document. For parts, it can mean 20+ material and 30+ color styles in every file. It consumes memory though only one color and one material is used at a time. Multiplied by the hundreds of unique parts in a large assembly, it adds up quickly. By moving styles into the external style library, only in-use styles are cached in the document's memory space, reducing the memory footprint of each document.

Style Libraries adapt to your unique needs.

Organizations can have a single, global library used by all designers, or each project can have its own specific library. This flexibility, along with a well-defined set of document templates, makes handling and encapsulation of project data manageable.

What is stored in a style library?

Autodesk Inventor provides a style library that contains a set of common materials, colors, and lighting for parts and assemblies, as well as common drafting standards such as ANSI, ISO, and DIN for drawings. Sheet metal styles include gage or sheet thickness as well as definitions for bends, corners, alternative flat pattern representations and unfolding preferences. Style libraries are a common source of styles for all documents in a design project. Each style library is a collection of .xml files, one for each style type. For example, there is a file for balloons, dimensions, layers, parts list, and all other style types.

The location of a Style Library is limited by one factor. Only one Style Library can exist in a directory. Style Libraries are stored on any local or network location.

When to use style libraries

A style library is a good way to manage styles so that they can be shared across documents in a design project. For example, material styles can be made available to designers so that all parts and weldments have access to the same definitions. Commonly used sheet metal materials, gauge or sheet thicknesses, and manufacturing preferences for corners and bends can be shared and easily accessed. In drawings, using styles assures uniform annotations, including such details as arrowheads, dimensions, leaders, center marks, and so on.

A good time to start using style libraries is when you start a new project, are new to Autodesk Inventor, or are an individual designer or work in a small group. You can use the default style library associated with the drafting standard.

To get started, you can use the default styles that are associated with each drafting standard. That is, ANSI comes with its own set of styles, as does DIN, JIS and other international standards. As your company standardizes on how to format objects, styles can be added or edited, and the changed styles saved to the style library. New documents will automatically have the styles available for formatting.

You might want to transition to using a style library later if you have active legacy projects, you work in a large workgroup, require PDM control of projects, and have need for one or more custom style libraries.

Use a style library when:

- You work with large assemblies and need a good way to share style information among multiple documents.
- You work in a workgroup that needs a common source of styles.

- You use document formatting changes from a CAD Administrator.
- Need to easily deploy changes to styles.
- You share formatting standards without providing internally controlled templates or documents.

Consider using a Style Library out of the box if you:

- Are starting a new project or are a new user.
- Work as an individual or in a small workgroup.
- Can probably use the Default Style Library that shipped with Autodesk Inventor.

Working without a style library

The project file (.ipj) includes a Use Style Library option that turns a style library on or off. When you turn a style library off, styles are only available in a document template. (It is the default for all pre-Autodesk Inventor 9 projects). If you change styles in the templates, all future documents will access the changes, but previously created documents will not.

Not using a Style Library has the following disadvantages:

- Updating documents to consume style changes from a CAD Administrator and sharing styles between documents is more difficult. Styles must be manually exported, and then imported into every document requiring the change, including all template files for a project.
- There are no capacity gains for large assemblies because all required style definitions are stored in every document.

If you later decide to work with the style library on, update the styles for previously created documents.

To transfer styles from one document to another, import or export styles.

Working with a style library

There are two ways to specify the style library to use. You can:

- Set up a style library that is specific to a project. In the Project Editor, turn on the style library on, and then set the project's Style Library folder option. Browse to a directory that contains the desired style library. Its styles are available to all designers who use the project.

- Specify the default style library on the File tab of the Application Options dialog box. If the Style Library folder option of the project is set to Default, this default setting is used.

The default Style Library is located in the Autodesk Inventor Design Data directory. This directory is defined by the Design Data path and is found and changed by selecting Tools tab ➤ Options panel ➤ Application Options on the ribbon. The option is found on the File tab.

NOTE If the Design Data directory location changes (from the Application Options setting), all projects set to [Default] for their Style Library change to use the Style Library located in the new Design Data directory. This change does not affect objects inside the documents unless they find a matching style in the new library and updated manually to match.

When a project is created, or its Use Style Library option is changed from No to either Yes or Read Only, the project uses the default Style Library. It is indicated by [Default] as the setting for the Style Library folder option in the project file. The default Style Library is ideal for users who want to use a single style library for all of their projects.

The location of the Templates folder is by default:

Microsoft Windows XP: Program Files\Autodesk\Inventor [version]\Templates.

Microsoft Vista: Users\Public\Documents\Autodesk\Inventor [version]\Templates.

It may be appropriate to have a Style Library that is specific to each project. For example, a contractor may need a different set of styles to match each customer's formatting requirements for documentation.

A project is set to use a specific style library by setting the Style Library folder option to a specific path. If a Style Library exists in the specified directory, the project uses that Style Library. If the directory is empty, Autodesk Inventor creates an empty Style Library for the project.

Setting style library use for projects

Style Library use is determined by the active project file. There are two elements that define the setup:

- Will the project use a Style Library? This is determined by the project's Use Style Library setting. This also influences how document templates are used in a project.

- Which Style Library is used? Each project has the option to use any Style Library a user creates, but projects use only one Style Library at a time. The project Style Library Folder Option determines the location of the Style Library.

Use Style Library = Yes

If Use Style Library is set to Yes, the project uses the Style Library defined in the Style Library folder options. All automatic management of styles is performed on all files opened under the project. With this setting, you have full read/write access to the styles in the library, that means you can edit and change the styles in the library by performing a Save to Style Library from an Autodesk Inventor document. This setting gives users all the benefits of using a Style Library, including full sharing capability, streamlined updates of styles, and any potential capacity gain for large assemblies.

NOTE A Style Library is a group of predefined XML files. However, a Style Library is treated as a single entity by Autodesk Inventor. When a Save to Style Library is performed, the entire library is saved from the Autodesk Inventor internal memory back to disk. Use Style Library = Yes is recommended for CAD Administrators or by users who do not share Style Libraries with other users.

Use Style Library = Read Only

Setting a project's Use Style Library option to Read Only behaves like Use Style Library = Yes with one difference. You cannot change the Style Library using any of the Save to Style Library functions inside Autodesk Inventor.

NOTE If one of the XML files that make up a Style Library is set to Read Only through file properties, Autodesk Inventor treats this if the Use Style Library option was set to Read Only, even if Use Style Library = Yes. In this case, the entire library is treated as read-only even if some of the XML files are read/write.

Use Style Library = No

Setting a project's Use Style Library option to No prevents the project from using a Style Library. Required style definitions must then be loaded into the document in one of the following ways:

- Created in the document using the Style and Standard Editor.
- Loaded into the document using the Style and Standard Editor Import function.
- Be in the template document when the document is created.

Switching Between Using and Not Using a Style Library

Persistent use of a Style Library is recommended throughout the life of a project. As documents do not store unused styles that are in the project's Style Library, switching a project to not use a Style Library may leave your documents without the styles you expect to be available.

How do the templates and styles interact?

Depending on the Use Style Library setting, Autodesk Inventor templates take on a slightly different role:

If you set Use Style Library = No in the project, templates interact with styles in the same way as pre-Autodesk Inventor 9 templates. Styles are stored in the templates and are copied to new documents when created. If a style in a template is changed, that change does not propagate to existing documents based on that template. Required document styles must be added to the template files.

If you set Use Style Library = Yes or Read Only, templates are not the primary source of styles for new documents. They are the source of the default styles the document uses. When a new document is created, any styles in the template are copied from the Style Library as part of the document creation process.

When using a Style Library, styles that are in the template are refreshed to match the style library when a new document is created. If a style in a template does not exist in the library, it is added to the new document. If you want to have a specific set of styles you use all the time and you do not want all users to share a Style Library, we recommend you to use templates this way. For example, your company may have a single Style Library that all users share, containing all company-approved materials, drafting standards, and so on. However, a specific project may require a specific set of layers that are not part of the company standard. Instead of creating a separate Style Library for the project, the layers are added to the drawing template for the project.

TIP When setting up templates for a project using a style library, we recommended that you purge all styles that are not required in the project.

Save styles to a style library

You can create or edit a style, and then save the changed version to the style library. If the style has the same name as a style that exists in the style library, it replaces the library version.

Access to a style library may be controlled by the CAD administrator. It is usually a good practice for the CAD administrator to maintain a master copy

of a style library and copy it if modifications are needed. Using this practice, you can always retrieve the default styles from the master library.

- 1 Create or open a document with the styles you want to copy to the style library.
- 2 On the ribbon, click Manage tab ➤ Styles and Standards panel ➤ Save.
- 3 In the dialog box, all styles in the document are listed. Click column headers to sort contents.
 - In the Changed column, all styles that do not match the library are labeled Not in Library.
 - In the Save to Library column, styles that do not match the library are automatically labeled Yes. Click to change to No, if desired.
 - To mark all styles at once, click Yes to All or No to All.
- 4 Click OK to save styles marked Yes, or Cancel to close the dialog box without saving styles.

NOTE When you save a style to a style library, substyles are automatically saved. You are prompted to save referenced substyles that are different from the library. For example, a dimension style uses a text style as a substyle. If you change the dimension style and text style, saving the dimension style to the style library prompts you to also save the text style to the library.

Manage style libraries

Style Library Manager is a stand-alone tool used for the administration of style libraries. This tool works directly with the style library XML files on disk, and is not used in the management of styles inside Autodesk Inventor documents. It is accessed from the Microsoft® Windows® Start menu, and is found in the Tools submenu for Autodesk Inventor.

Use Style Library Manager to:

- Create new style libraries
- Manage styles in style libraries
- Manage styles between multiple style libraries

Create style libraries

Autodesk Inventor installs with a default style library designed to leverage of all new capabilities of styles.

You can create your own style libraries in two different ways: Copy an existing style library or create an empty style library. The target directory for the new library cannot include other style library.

After you create a style library, use Style Library Manager or Style and Standard Editor to add or remove particular styles.

Rename and delete styles inside of a library

Use Style Library Manager to rename styles inside of a library. Renaming a style in the library does not rename the same style in any other library or document. Once a style is renamed, all document copies of the style lose their name match and are considered locally cached style.

The only way to delete a style from a library is through the Style Library Manager.

NOTE Rename and Delete commands are accessed from the right-click menu when selecting one or more styles.

TIP There is no Undo in the Style Library Manager. Renaming, copying, or deleting of styles cannot be undone. We recommend you create a backup copy of your library before you make changes.

Copy styles between libraries

Styles are copied between libraries using the Style Library Manager. To copy a style from one library to another, load the source library into Style Library Manager and click the Copy Selected Styles command between the Style Library 1 and Style Library 2 panes. See the Style Library Manager Help for more details.

Manage Style Library Contents

A style library is a group of named XML files in a directory. There is one XML file per style type (for example: color.xml, text.xml). Since the names of the XML files must be the same for each style library, only one library can exist in a directory.

To be considered a valid style library, all predefined XML files must be present in the directory.

The best way to share a subset of styles from a library (for example, a single standard or all materials from a specific library) is to use the Import/Export styles function in the Style and Standard Editor, or create an empty style library and copy the desired styles using the Style Library Manager.

NOTE An exported style file (*.styxml) cannot be used as part of a style library.

Incomplete Style Libraries

Though a style library is made up of several separate XML files, Autodesk Inventor and the Style Library Manager treat the library as a single entity. If some of the predefined XML files required for a style library are missing from a directory, Autodesk Inventor displays an error message indicating files are missing and cannot be loaded.

Restoring the Autodesk Inventor Default Style Library

A utility for restoring the default Design Data directory files is located on the Autodesk Inventor Install CD in the Support directory. This utility replaces the original default style library that ships with Autodesk Inventor, as well as the Thread.xls and Clearance.xls spreadsheets.

Prevent Style Library Changes

In a multi-user environment, CAD administrators may want to prevent users from changing a style library. Set the library to Read-only. Two methods are available to make a style library read-only:

- Set the project Use Style Library option to Read Only.
- Use a Microsoft Windows file property setting. In Microsoft Windows Explorer, select all the XML files that make up the library or the library directory folder itself, right-click, select Properties, and set to Read Only.

In Autodesk Inventor, both methods are treated identically when accessing style libraries. If a single XML file is read-only, the entire library is treated as read-only.

An administrator can have two versions of a project file. One is for users (Use Style Library = Read Only), which prevents changes, and one is for administrators (Use Style Library = Yes), to actively edit the library without dealing with the individual Windows file properties of each XML file. For many workgroup setups, this may be sufficient for management of style libraries.

NOTE The project file option has the disadvantage that any interaction done outside Autodesk Inventor treats the style library as read/write. This includes the Style Library Manager, which works with library files outside the context of an Autodesk Inventor project. The Style Library Manager only references project files as a shortcut to the XML files.

Style Libraries and Autodesk Vault

Style libraries are not directly supported by Autodesk® Vault. Style libraries are manually vaulted and managed using the Vault Explorer.

NOTE

- Get, Check in, and Check Out must be done through the Vault Explorer.
 - Vaulted Style Library XML files are set to read-only through the file properties, just as any standard Autodesk Inventor file or any other 3rd party file that is vaulted manually.
 - If editing the style library is required, all XML files must be checked out.
-

Manage multiple style libraries

To simplify management of multiple style libraries, consider the following:

- Keep all working style libraries as read-only to prevent accidental changes.
- Use a parent/child style library scheme:
 - The parent is a library unused by any active project.
 - All project style libraries are considered children.
 - The administrator makes all changes in the parent version.
 - Push changes to a child by copying the styles from the parent version using the Style Library Manager.
- Try to reuse styles if possible.
- Most users utilize the same material, color, and lighting styles, even if they require a different set of drawing-related styles.
- For management of drawing related styles, give the primary standard and its object defaults unique names for each project. Reuse dimensions, text, and layers as much as possible.

Pack and Go

The Autodesk Inventor Pack and Go utility optionally includes a style library. By default, a project's style library is included in the target directory created by Pack and Go. This is true even if the default library is used, because the recipient's default library may differ from the one used by your project(s).

The packed style library is included in the Pack and Go project in a subdirectory called Design Data. The Pack and Go project file has the Use Style Library setting the same as the original project, but its style library folder option is set explicitly to the Pack and Go location of the packed up style library.

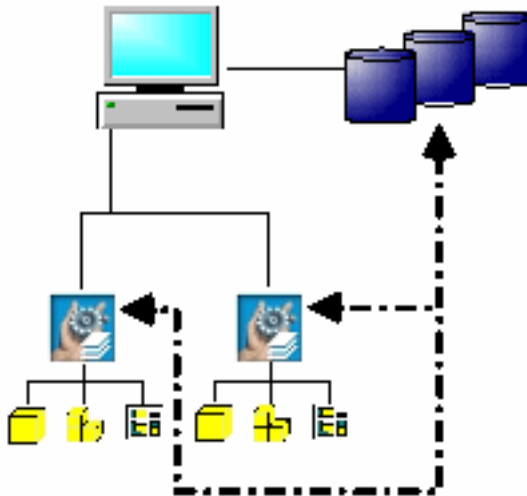
Configure style libraries

Style Libraries can be stored on any local or network location, but only one Style Library can exist in a directory.

Select the Style Library configuration that best corresponds with your needs:

Individual User - Global Style Library

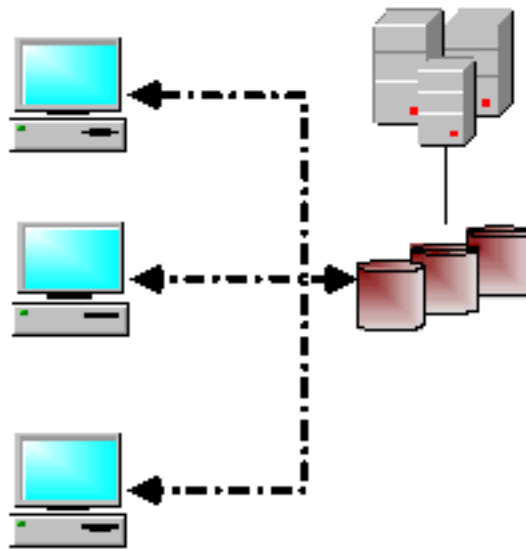
For individual users who do not need multiple project-specific styles, a single, global style library may be the best setup. Users have a single Style Library that is in the Design Data directory on their local drive, and all projects have the Style Library folder option set to [Default].



Multiple Users - Global Style Library

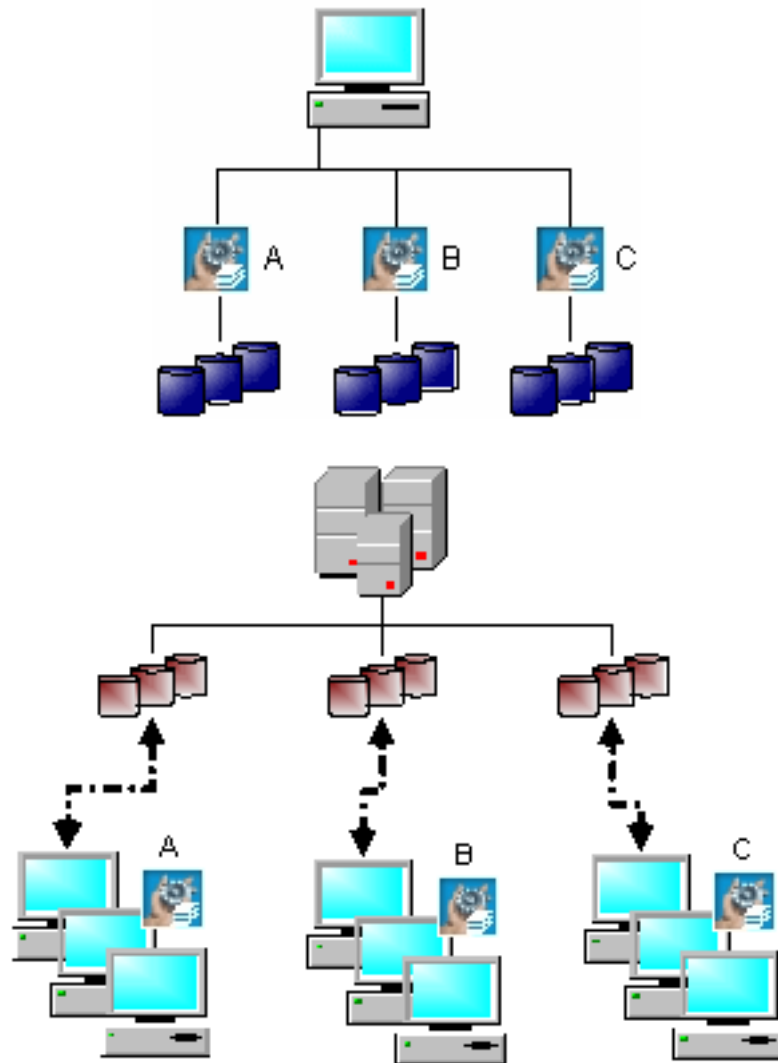
For workgroups that require a single style library for all projects, a Style Library should be located on a common server. Each user's Design Data directory points to this server location, and every project has the Style Library folder option set to [Default]

NOTE In addition to the Default Style Library, the Thread.xls and Clearance.xls spreadsheets used by the Autodesk Inventor Hole feature command are located in the Design Data directory. If the Design Data directory is changed, be sure to copy the Thread.xls and Clearance.xls spreadsheets files into this new directory.



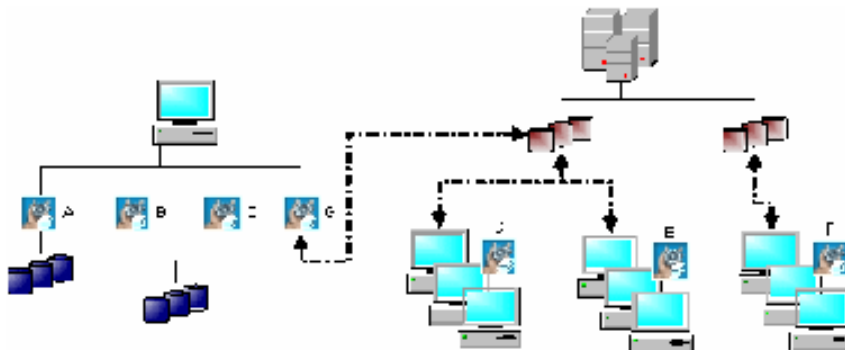
Project-Specific Style Libraries

For users who need a project-specific Style Library, the recommended best practice is to place the library in a subdirectory of the project directory. For workgroups, Style Libraries are best placed on a common server in separate directories. The libraries can be manually administered using Autodesk® Vault, and copied onto local computers in a manner like other Autodesk Inventor files.



Mixed Scheme Setups

There are no restrictions on how users set up their project/style library schemes. Any project can use any style library.



Create a style library

You can create a style library using one of several methods. In general, new style libraries are created by CAD administrators so that uniform styles are assured throughout a design project.

Individual design projects can use a custom style library, which is specified in the project file. Because it is specified in the project, all designers who use the project use the same set of styles for formatting.

TIP It is a good idea to keep the original style libraries associated with drafting standards intact. Instead, make a copy of the master library and use it to customize style libraries for individual design projects.

- 1 Click Start menu ► Programs ► Autodesk ► Autodesk Inventor ► Tools ► Style Library Manager.
- 2 Click the Create New Style Library command above the Style Library 2 pane.
- 3 In the Creation Method box, click the arrow to select Copy Existing Style Library or Create Empty Style Library.
- 4 In the New Style Library Location box, accept the default location or browse to the folder where you want to store the new library. If the library is accessed by a project, place it in a folder recognized by project locations.
- 5 In the New Style Library Location box, accept the default location or browse to the folder where you want to store the new library. If the library is accessed by a document, place it in a folder recognized by design file locations.

- 6 If copying a style library, in the Source Style Library to Copy box, accept the default library or browse to the library you want to copy. This option is unavailable if you create an empty library.
- 7 Click OK to create the library and close the dialog box.

Exit the Style Library Manager. Use the Style and Standard Editor to customize the styles in the new style library.

Copy, rename, or delete library styles

The Style Library Manager is primarily used by CAD administrators to copy, rename, and delete library styles. The CAD administrator controls the style library contents and assures that designers using styles have the same style definitions, assuring uniform document formatting.

The Style Library Manager uses the Create New Style Library command to create new style libraries. Create New Style Library is also available from the Style Management Wizard.

In general, style libraries are read-only, except to make necessary changes. The Styles Library Manager uses the read-only file property, and disregards the library status in a project file (.ipj), specified by the Use Style Library=Read Only option.

Any changes made to a style library are not available in other documents until the current Autodesk Inventor session closes and a new session is opened.

TIP It is a good practice to maintain a master copy of a style library, and then copy it before you modify for project-specific style libraries.

You can create new styles or rename styles in a document, but they affect only that document. Take a separate step to save a new or changed style to a style library to replace the library version. Access to the style library may be controlled by the CAD administrator.


Copy styles from one style library to another style library

You can develop style libraries that are specific to a given project. This procedure copies styles you have already defined in one style library to another style library.

- 1 Click Start menu ► Programs ► Autodesk ► Autodesk Inventor ► Tools ► Style Library Manager.

- 2 In Styles Library 1, click the down arrow to select or click Browse to locate the style library whose styles you want to copy.
- 3 In the Choose Style Type column, click a style type to list all defined styles in Style Library 1.
- 4 In Style Library 2, click the down arrow to select or click Browse to locate a style library.
If appropriate, click Create New Style Library:
 - In Creation Method, select Copy Existing Style Library or Create Empty Style Library.
 - Browse to the folder where the library is located.
 - If copying a style library, click the arrow or browse to select the Source Library for Copy. The existing styles are listed in the Style Library 2 window.



- 5 Click one or more styles, and then click  to add them to Style Library 2.
- 6 Continue to click style types to add to Style Library 1, and then select and add them to Style Library 2.
- 7 Styles in one library might not match the styles in another library. Compare the styles in both libraries.



Click **Show All Styles** to list all styles in both style libraries.



Click **Show Mismatched Styles** to show styles whose definitions are different in the two libraries. Mismatched styles are shown in red.



Click **Show Unique Styles** to show styles that exist in one library but not the other. Unique styles are shown in blue.

- 8 Click the right and left arrows to add or remove styles as desired.
If mismatched substyles are detected, the Manage Styles dialog box opens. Indicate to overwrite the version of the substyle in the destination library or retain the current version.
- 9 Click Close to save the libraries.

NOTE Only the selected style is copied from one Style Library to another. Substyles are only copied if they do not exist in the destination library. If a style exists with the same name, the destination version is used.

Rename styles in a style library

A style name can change in the style library, but all documents that reference the old name lose their association to that style.

- 1 Click Start menu ► Programs ► Autodesk ► Autodesk Inventor ► Tools ► Style Library Manager.
- 2 Click the arrow or browse to the library whose style you want to rename.
- 3 In the Style Library 1 or 2 window, right-click a style name and select Rename.
A **warning** asks you to confirm that documents links to the style will be broken. Click Yes to continue or No to cancel renaming. If Yes, enter the new style name.
- 4 Continue renaming as needed, and then click Close to save the changes.

Delete styles from a style library

A style can be deleted from a style library, but all documents that reference the old name lose their association to that style.

- 1 Click Start menu ► Programs ► Autodesk ► Autodesk Inventor ► Tools ► Style Library Manager.
- 2 In the Style Library 1 or 2 window, click the arrow or browse to the library whose styles you want to delete.
- 3 Select a style type in the Choose Style Type list. Then right-click a style name in the Style Library 1 or 2 window and select Delete.
A warning asks you to confirm that the style will be permanently deleted from the style library. Click Yes to delete the style or No to cancel delete.

NOTE You cannot use Undo to reverse a deletion.

- 4 Continue deleting as needed. When finished, click Exit to close the Style Library Manager dialog box.

TIP Use Manage tab ➤ Styles and Standards panel ➤ Purge to remove unused styles from the current document, and optionally, all referenced documents.

Harvest or purge styles in a batch process

The Style Management Wizard is a tool for administrators to assist in batch-style management for Autodesk Inventor files. Batch processes harvest or purge styles for multiple files in one operation. All files must migrate to the most recent version of Autodesk Inventor or they are skipped during batch processing, including assemblies that were migrated but contain components that were not migrated.

To open the Style Management Wizard, click Start menu ➤ Programs ➤ Autodesk ➤ Autodesk Inventor ➤ Tools ➤ Style Management Wizard.

Harvest styles in a batch process

Specify a group of Autodesk Inventor files (such as parts, assemblies, and drawings) and a target style library. Using the Style Management Wizard, all styles from the selected documents are added to the style library.

This process is useful if you did not make extensive use of templates for the source of styles, but primarily created styles on-the-fly in individual documents.

- 1 Select a project to manage. In the listed projects, double-click to make a selection. The Style Library settings and Autodesk Inventor version number are listed below the window. Click Next.
- 2 Select files to process. Add files using one of these methods:
 - **Add Specific Files command** Click to open the Choose Files to Process dialog box. Browse to the folder that contains the files to process. Select one or more files and then click Open. Selected files are listed by file name, file type, and path.
 - **Add All Files in Active Command** Lists all files accessible in project folders file name, type, and path. Clear the check box to exclude a file from processing.

If you prefer, use an alternate method to add files to the list:

- **Drag and drop files** from Windows Explorer. Drop them in the list to add them.
- **Get all files referenced by a specific file** by right-clicking the file in the process list, and then selecting Get Referenced Files. Adds all files that the selected file is dependent on to the list.

- **Get all drawings or presentations that use a specific file** by right-clicking a file in the process list. Select Drawings and Presentations, and then Get All. Adds to the process list all drawings and presentations in the project search paths that use the specific file.

TIP Click the column headings of the files list to sort the window contents.

- 3 On the Select Management Options page, select Harvest Styles into Target Style Library.
- 4 Select the target library, the destination of the harvested styles. Choose one:
 - **Use Source Project Style Library** automatically selects the style library specified in the project.
 - **Create a New Style Library** opens the New Style Library dialog box. Specify to copy an existing style library or create an empty library. Browse to the folder where the library will be stored, and if copying, the source style library you are copying.
 - **Select an Existing Style Library** lists the Default Style Library, but you can browse to a different library.
- 5 Click Next.
- 6 If necessary, review the selections you have made, and then click Start to harvest styles to the specified library.

A log file is generated during processing. You can press Pause to halt the processing temporarily or Cancel to stop.

Purge styles in a batch process

Specify a group of Autodesk Inventor files (such as parts, assemblies, and drawings) and remove every unused style in all documents. This technique is a fast way to remove styles from legacy documents that you no longer need after you start using a style library.

NOTE Use caution when purging styles. Every unused style is removed permanently from all processed documents. If the styles are not saved in another document or a style library, they are permanently lost. Always back up your data before purging styles.

- 1 Select a project to manage. In the listed projects, double-click to make selection. The Style Library settings and Autodesk Inventor version number are listed below the window. Click Next.

NOTE The source project must have a style library.

- 2 Select files to process. Click one:
 - **Add Specific Files command** Click to open the Choose Files to Process dialog box. Browse to the folder that contains the files to process. Select one or more files, and then click Open. Selected files are listed by file name, file type, and path.
 - **Add All Files in Active Project command** Lists all files accessible in project folders file name, type, and path. Clear the check box to exclude a file from processing.

If you prefer, use an alternate method to add files to the list:

- **Drag and drop files** from Microsoft Windows Explorer. Drop them in the list to add them.
- **Get all files referenced by a specific file** by right-clicking the file in the process list, and then selecting Get Referenced Files. Adds all files that the selected file is dependent on to the list.
- **Get all drawings or presentations that use a specific file** by right-clicking a file in the process list. Select Drawings and Presentations, and then Get All. Adds to the process list all drawings and presentations in the project search paths that use the specific file.

TIP Click the column headings of the files list to sort the window contents.

- 3 On the Select Management Options page, select Purge All Unused Styles from Files.

Be sure that you want to purge all unused styles from all documents. Styles not saved in a style library are permanently lost. Click Next.
- 4 If necessary, review the selections you have made, and then click Start to purge all unused styles from all selected documents.

You can press Pause to halt temporarily the processing or Cancel to stop.

Quick Reference

Save styles to style library

Saves selected styles in the document to the style library. The current style library version is replaced with the new version.

In other documents that use the style, you must manually update to apply the new version of the style in the library.

Access:



Ribbon: Manage tab ► Styles and Standards panel ► Save

Click a column heading to sort contents. All columns resort to correspond with the new order.

Document Name	Shows file name in which the style is used.
Style Name	Lists styles in the current document.
Type	Identifies the style type.
Changes	Indicates if a style was changed in the document or library. If the style does not match the library, it is marked Not in Library.
Save to Library?	Switches individual styles to save to the library. If Yes, the style in the library is updated with the version in the current document. If No, the library style is not updated with the changed style in the current document.
<hr/> NOTE Styles that are not in the library are automatically switched to Yes, but can be changed to No. <hr/>	

Click **Yes to All** to save all styles in the document to the style library or **No to All** to retain the styles in the current document, but not update the library.

NOTE When you save a style to a style library, substyles are automatically saved. You are prompted to save referenced substyles that are different from the library. For example, a dimension style uses a text style as a substyle. If you change the dimension style and text style, saving the dimension style to the style library prompts you to also save the text style to the library.

Quick Reference

Create New Style Library

Creates a style library by specifying the creation method, the library location, and the source library.

Access:

Click Start menu ► Programs ► Autodesk ► Autodesk Inventor ► Tools ► Style Library

Manager. In the Style Library Manager, click Create New Style Library.



In the Create New Style Library dialog box:

Creation Method selects how you want to create the library. Click the arrow to choose Copy Existing Style Library, and then modify as appropriate or Create Empty Style Library, and then select styles from one or more libraries.

New Style Library Location browses to the folder where you want to store the new library.

Source Style Library to Copy browses to the library you want to copy when the creation method is Copy. Unavailable when the creation method creates an empty style library.

Quick Reference

Style Library Manager

The Style Library Manager is used primarily by a CAD administrator to:

- Copy styles from one style library to another style library.
- Rename styles in a style library.
- Delete styles from a style library.

Any changes made to a style library are not available in other documents until the current Autodesk Inventor session closes and a new session is reopened.

Some styles reference other styles as substyles (for example, the dimension style references the text style as a substyle). When copying a style library, the Style Library Manager opens the Manage Styles dialog box if mismatched substyles are detected in the source and destination libraries. For each substyle, confirm to replace the current version with the source version.

NOTE If the source library substyles are not copied, substyles from the destination style library are used instead. The overall definition of the dimension style changes, because the set of substyles it uses has different characteristics compared to the source.

The Style Library Manager also accesses the Create New Style Library dialog box to create new style libraries. The same dialog box is also accessible from the Style Management Wizard.

Access: Click Start menu ► Programs ► Autodesk ► Autodesk Inventor ► Tools ► Style Library Manager. In the Style Library Manager dialog box, click a style type to populate the Style Library pane with defined styles.

Choose Style Type

Lists all style types. Click a style type to show all definitions in the Style Library column.

Style Library 1

Shows styles of the selected type in the specified library.

- Arrow** Lists projects that have style libraries associated with them. Click to select.
- Browse** Click to browse to the appropriate style library.

Style Library 2

Shows styles of the selected type in the specified library.

- Arrow** Lists available style libraries. Click to select.
- Create New Style Library** Creates a style library by copying an existing library or creating an empty library, specifies its location, and names the source library to copy, if applicable.
- Browse** Click to browse to the appropriate style library.

Comparing Styles Between Definitions

If a style exists in both Style Libraries and is an exact name value match (that is, Equal), then the text in the style lists has a normal font display.

If a style exists in both Style Libraries, is an exact name match, and is not a value match (that is, Not Equal), then the text for this style name displays as **bold, underlined, and red**.

If a style only exists in one Style Library (that is, is Unique), then the text for this style name displays as **bold, underline, and blue**.

Filter commands

Shows differences between the lists in the style libraries:



Click **Show All Styles** to list all styles in both style libraries.



Click **Show Mismatched Styles** to show styles whose definitions are different in the two libraries.



Click **Show Unique Styles** to show styles that exist in one library but not the other.

Copy style commands



Copies selected styles from Style Library 1 to Style Library 2.



Copies selected styles from Style Library 2 to Style Library 1.

Substyles are only copied from one style library to another if they do not exist in the target library. To avoid replacing an existing style accidentally, you are prompted to replace the target style with the source style. It occurs if a substyle exists in the target style library, and it is different from the substyle in the source library.

Manage styles dialog box

When copying a style library, if a source style and one or more of its substyles differ from the destination library, you are asked to select which substyles to copy to the destination library.

For example, a dimension style uses the text style as a substyle. If the text substyle referenced by the dimension style is different from the text style in the destination style library, confirm to replace the destination text style with the text substyle from the source style library.

If you choose to use a combination of Yes and No in the Overwrite column, the style in the destination style library is not identical characteristics compared to the source.

Access: In the Style Library Manager dialog box, click the Show Unique Styles command or Show Mismatched Styles command to list styles in red text. Listed styles do not match in the source and destination style libraries. Right-click a red style name and then click the arrow button to copy it. The Manage Styles dialog box opens.

Style Name	Lists the destination style name and its standard.
Path	Shows the path of the destination style.
Type	Lists the style type.
Overwrite?	The default setting is No. For each listed style, click in the column to switch to Yes.

To set all styles at once, click Yes to All (to overwrite styles in the destination library) or No to All (to copy only the parent style, but not the substyles).

Quick Reference

Style Management Wizard

The Style Management Wizard is a tool for administrators to assist in batch-style management for Autodesk Inventor files. Using the Style Management Wizard, you can:

- Harvest styles from Autodesk Inventor files and place them in a target style library.
- Purge styles from Autodesk Inventor files. Purging removes unused style information and can be beneficial in large assemblies by reducing required memory.

Use the Style Management Wizard when transitioning to using a style library with your design project.

Access: Click Start menu ► Programs ► Autodesk ► Autodesk Inventor [version] ► Tools ► Style Management Wizard.

All files must migrate to the latest version of Autodesk Inventor. Files that are not migrated are skipped during batch processing, including assemblies that are migrated but contain components that are not migrated.

Welcome page	Describes the Harvest Styles and Purge Styles processes.
Projects to Manage page	Lists all defined projects by Name and Project Location. The active project is indicated by a check mark. Its settings for Style Library and path are summarized below the window. Double-click to select a project and make it active. The Style Management Wizard uses the active project to resolve all file paths during processing.
Files to Process	<p>Selects files to process in a detailed view.</p> <p>Add Specific Files command Browses to the folder that contains the files to process. Selected files are listed by file name, file type, and path.</p> <p>Add All Files in Active Project command Lists all files by file name, type, and path. Clear the check box to exclude a file from processing.</p> <p>Alternate methods of adding files to the list:</p> <ul style="list-style-type: none">■ Drag and drop files from Microsoft Windows Explorer. Drop them in the list to add them.■ Get all files referenced by a specific file by right-clicking the file in the process list, and then selecting Get Referenced Files. Adds all files that the selected file is dependent on to the list.■ Get all drawings or presentations that use a specific file by right-clicking a file in the process list. Select Drawings and Presentations, and then Get All. Adds to the process list all drawings and presentations in the project search paths that use the specific file.
Select Management Options page	<p>Chooses a harvest or purge operation.</p> <p>Harvest Styles into Target Style Library specifies the target style library.</p> <p>Use Source Project Style Library Automatically selects the style library specified in the project. Clear the check box of files you want to exclude from processing.</p>

Create a New Style Library Opens the New Style Library dialog box. Specify to copy an existing style library or create an empty library. Browse to the folder where the library will be stored, and if copying, the source style library you are copying.

Select an Existing Style Library Lists the Default Style Library and any project that uses a style library. Use the Browse to browse to a specific library.

Purge All Unused Styles from Files automatically deletes unused styles from selected documents. Styles not saved to a styles library are permanently lost.

Begin Batch Processing Starts batch processing, using previously selected options. Press Pause to halt the processing temporarily or Cancel to stop.

Configure the project (.ipj) file using the Project Editor

A project file defines the locations of all files associated with a project, including:

- where design data are stored.
- where you edit files.
- how many versions of the file are retained when you save a file.
- Content Center configuration settings.
- the project type.

Before creating your first projects, set up your folder structure.

For simplicity (and present or future Autodesk Vault compatibility), we recommend to use a single project file. A single project file does not contain any other project file and is used to control all designs. With the single project file configuration you gain:

- **Simplicity:** With a single project, you use the same project file for every design in your environment. You do not have to remember what project file to use with which data set. You also benefit from having a single common location for all your data.

- **Reduced Resolution Failures:** Inventor can automatically search for any missing files throughout your entire data set. This practice greatly reduces the number of times you must manually locate missing files.
- **Increased Design Reuse:** Simplify the reuse of other data from your current designs or vault.

This method prevents you from adding a library path or copying a file from another project.

Set up a folder structure for your project before you create it and start a design. Base a project plan on the existing and future files associated with the project.

There are two types of Project files: the Vault project file (intended for a shared workgroup environment where users share files) and the single user project file (intended for users not working in a workgroup).

We recommended CAD Managers create an Autodesk Vault project file to manage files in a shared environment. Users can check out a file and work on it in their individual local workspace. It also simplifies the process of renaming files and copying existing designs.

Some guidelines for the folder structure are:

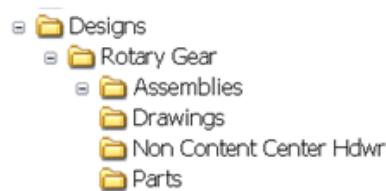
Follow your company standards and naming conventions for the project folders.

Set up a main folder for them if you plan to use existing files.

Set up subfolders under the project folder (workgroup or workspace).

Create a project before you start your design work so that the file referencing information is stored appropriately and reduces the possibility of file resolution problems.

For example, create your project at the Designs level:



Plan the project:

- Determine whether the project files and libraries are located locally or on a network.

- Determine which project type most closely fits the work situation for the particular project: vault, semi-isolated master, semi-isolated workspace, shared, or single-user. The correct project type is set automatically when you create the project file
- Use the Project wizard to create the project in accordance with your project plan.
- Use the Project Editor to set additional options, if needed.
- Configure your project to include only libraries you use. Remove all unused Content Center libraries to reduce the amount of displayed data and to increase performance. (Libraries removed from the project remain on the server, and can be added to the project later).
- Create a common project with one or more library paths. Place the project file on a shared location. Have each team member create a personal project, using the common project as a template.

For more information, refer to the online help in Autodesk Inventor:

- 1 In Autodesk Inventor, click Help.
- 2 Select the Contents tab, and navigate to the topics:
- 3 Autodesk Inventor ► Collaboration and Visualization ► Design Assistant ► Projects and Libraries: [Learn about projects](#) on page 73

IMPORTANT Remember to click **Back** to return to the CAD Manager Guide if you click any of the links on this page.

Projects and libraries

Learn about projects

Autodesk Inventor uses projects to identify folders that contain files unique to a design project and folders that contain library files referenced by a design project. Projects define the locations of templates, styles, Content Center configurations, and the vault mapping of the project.

Use the Project wizard to define a project for each design project so that you have access to design files and libraries and can maintain file references. After

you create a project, use the Project Editor to set options such as the number of file versions to keep when you save files and the location of frequently used subfolders.

A typical project consists of parts and assemblies unique to the project, standard components that are unique to your company, and off-the-shelf components such as fasteners, fittings, or electrical components. Because projects cross-file references are stored relative to the project folder locations, you can maintain the references when you move, archive, and restructure the project folders by updating the project file data.

NOTE Projects always use relative paths rather than absolute paths when the project locations such as the workspace, workgroup, or libraries are in a subfolder of the folder that contains the project file.

TIP Before creating your first projects, set up your folder structure.

What is a project file?



A project file is a text file in .xml format with an .ipj extension. The file specifies the paths to the folder containing the files in the project. To assure that links between files work properly, add the locations for folders to the project file before working on model files. You can have as many projects as needed to manage your work.

The project shortcut is located in the projects folder. The project file (.ipj) location is specified in the Project wizard when the project is created.

What information is in a project file?

The project type designates either a shared location called a workgroup or a personal editing location called a workspace. Many projects also specify one or more libraries, locations where files you reference but do not edit are stored. Depending on the project type, a project file can contain the following sections:

Workspace

Specifies the location where you create, edit, and save files for vault, semi-isolated workspace, and single-user projects. In semi-isolated mode, a master project defines the shared location (workgroup) but does not contain a workspace.

Workgroup search paths	Shows the shared network locations defined in shared and semi-isolated master projects where Autodesk Inventor searches for referenced files specific to the project, after searching the workspace. Files in a workgroup cannot be edited until they are checked out. Only one designer can check out and edit a file at a time.
Libraries	Shows the locations for standard parts, iParts, or other named libraries. You can create library folders for any parts and assemblies that you reuse or share with other designers, but do not intend to edit.
Included path files	Specifies the path and name of another project file. In a semi-isolated workspace project, the semi-isolated Master project is the included file. The included project file specifies the location of workgroup files or libraries on a network. Only one project file can be included.
Frequently used sub-folders	Lists subfolders that are nested within a project folder structure. The listed folders appear as shortcuts in the Open, Save, and Place dialog boxes, so you can quickly navigate to the location. They are not used to resolve file locations. Shortcuts to frequently used subfolders are preferred rather than adding nested workgroups or workspaces.
Using unique file names	Specifies the intent to name files unique in a project.

How does a project file work?

The file locations in a project file work as search paths for Autodesk Inventor. When you open a file, Autodesk Inventor searches the locations specified in the active project file to find the file and any referenced files. The sections in the file set the search order. Within each section, the paths are searched in the order in which they appear.

NOTE If files with the same name exist in more than one search location, Autodesk Inventor opens the first file it finds.

How do I set up a project?

Set up a folder structure for your project before you create it and start a design. Base a project plan on the existing and future files associated with the project. Some guidelines for the folder structure are:

- Follow your company standards and naming conventions for the project folders.

- If you plan to use existing files, set up a main folder for them.
- Set up subfolders under the project folder (workgroup or workspace).

Create a project before you start your design work so that the file referencing information is stored appropriately and reduces the possibility of file resolution problems.

- 1 Plan the project.
 - Consider the types of existing and future files associated with the project.
 - Determine whether the project files and libraries are located locally or on a network.
 - Determine which project type most closely fits the work situation for the particular project: vault, semi-isolated master, semi-isolated workspace, shared, or single-user. The correct project type is set automatically when you create the project file.
- 2 Use the Project wizard to create the project in accordance with your project plan.
- 3 Use the Project Editor to set additional options, if desired.

NOTE In the Project wizard, Semi-isolated and Shared project types are unavailable by default. Use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy projects types, click Tools tab ➤ Options panel ➤ Application Options ➤ General tab. Select the "Enable creation of legacy project types" check box.

If the check box is not selected, and Autodesk Vault is not installed, only a Single User project can be created with the Project Wizard.

Use best practices in projects

- **Included Path**
Autodesk Vault does not allow the use of an Included Path. If possible do not use an Included Path. We recommend that you use Autodesk Vault to manage files.

NOTE If you use the Included path option, Set the Included file path option, if used at all, to point to a single, read-only project file, maintained on a network by the CAD administrator. The only defined project path that to include in the local project of each user file is the Workspace.

■ **Workspace**

We do not recommend that you have a Workspace on a network location. Perform all work on files on your local drive, and when finished, copy the files back to the network.

If you use a separate project file for each project and locate the files specific to that project, including the project file, in the workspace folder, copy the entire project folder back and forth between your local disk and the shared server. You can also copy the necessary or frequently used libraries to your computer, provided you have sufficient space. Pack and Go can be useful in creating a local copy of the libraries that contain only the library files referenced from your project.

Make the Workspace local to the computer of each user.

NOTE Autodesk Vault is recommended for managing files in a shared environment. Using Autodesk Vault to manage your files, ensures the files maintained on the server are copied over to the local Workspace when checked out.

Do not define storage locations that point to subfolders of the Workspace.

■ **Workgroups**

Avoid use of Workgroups altogether where possible.

The number of Workgroup search paths defined affects the time it takes to search. If your environment necessitates the use of Workgroups, limit the number as much as possible.

NOTE When Autodesk Inventor cannot find components immediately, it can result in a lengthy search and slow user interaction.

■ **Mapped Drives**

To reference a location on a server or remote workstation, use a Unique Naming Convention (UNC) path (such as: \\server_name\our_folders). Ensure that our_folder is shared with everyone who accesses the project location.

Avoid using different drive mappings or different UNC paths to the same file. The Open and Place Component dialogs allow you to navigate using arbitrary UNC paths to network locations. Avoid using this general

capability, and instead use only the project shortcuts in those dialogs to the folders in your project.

NOTE Using an alternative UNC or mapped drive path to navigate to a project location manually can confuse the Windows Operating system or the network domain mappings, causing it to drop the project based network mappings, resulting in an inability to locate referenced files.

Remove any mapped drives that no longer exist.

■ **Directory Structure**

Keep all of your projects under the projects folder. Organize and group them by subfolder as desired. Similarly, group all of your libraries under a common folder so they can be easy to find.

Within a given project, always have exactly one subfolder for each project. Have the project file in that subfolder and make that subfolder the project workspace. (The Project Editor defaults to create new projects according to this convention.) Locate all files that are specific to just one design under that workspace folder.

Never define a workspace or workgroup that refers to a folder in some other project.

Keep the directory structure under a given project location (i.e., workspace, library, and so on) relatively flat. The more folder structure that exists, the harder it is to maintain unique names for all files under the project non-library locations.

If there are hundreds of files in one folder, you may experience some performance slow downs and it makes it harder for you to work with them. A reasonable structure for projects that have more than 50-100 files might make one folder under the workspace for parts, another for subassemblies, another for drawings, and so on.

Place the top-level assemblies directly in the workspace folder.

Ensure that the Using Unique File Names option is set to true, and that every file under the workspace has a unique name. If you copy a file from one subfolder to another, make sure to rename it.

If you move files between subfolders, check the option under the Save tab on the Application Options dialog box to specify that reference changes must be saved. Then open and save all top-level assemblies and drawings. Make sure the path definitions in the .ipj file are correct.

■ **Styles Library**

Copy or move the style XML files from the Design Data folder to a shared location that can be accessed by all users. A new style library can alternatively be generated using the Style Library Manager utility.

NOTE This practice ensures the styles with textures and bitmaps, created from one project can be used in another project that uses the same styles library.

■ **Content Center Libraries**

We recommend that you share the same Content Center Library folder for all projects.

Create projects

Use projects to specify where design data is stored. The Project wizard steps you through specifying the project type, project name, location of the workgroup or workspace (depending on the project type), and names of one or more libraries.

Technical note - project format change

The format of the project file changed in R10, but pre-R10 versions of Autodesk Inventor can read R10 and later project files. Because of the file format change, pre-R10 Autodesk Inventor cannot detect usable information in the project file and replaces the file.

Create a folder for R10 and later Autodesk Inventor projects to isolate them from older project files. Set the Application Options ► File tab ► Projects Folder to a folder that has R10 in its name and that contains only R10 and later projects. For example, name the folder "C:\My Documents\Autodesk Inventor\R10."

When you save a new project in R10 or later, a backup copy of the project file is placed in the Old Versions folder, so you can recover it if it is accidentally replaced. When you migrate a pre-R10 project file, and Autodesk Inventor detects a project file with the same name in the Old Versions folder, you are asked to restore the R10 or later project and use it instead of the older project.

All project types have these default settings:

- Depending on the project type, one workgroup or one workspace is defined per project at the root level (".\"), and contains the project file (.ipj).
- The project shortcut is located in My Documents by default or a specified project folder location.


Create an Autodesk Vault project

We recommend that you use the Autodesk Vault project type to manage multi-user projects.

Install Autodesk Vault to create a vault project. The Vault project type includes the name and location of the vault, a specified workspace where you create and edit files, and one or more libraries. Create the vault project before you start any model files.

In a Vault project, workgroup search paths are not available. You consolidate all project folders under the single workspace search path. The workspace is the local folder mapped to the corresponding folder in the vault. The workspace can be a single folder, or can include a hierarchy of subfolders to help organize the project.



- 1 Click  ► Manage ► Projects.
- 2 In the Projects dialog box, click Projects, and then click New to start the Project wizard.
- 3 Select New Vault Project, and then click Next.
- 4 Name the vault project and specify the project location. Make sure it is a new folder that does not contain any data. By default, the Project wizard creates a folder for the project file (.ipj), but if you browse to a different location, the located folder name is used. Click Next.
- 5 Select existing libraries, add them to the project, and then click Finish.
- 6 In the Project Editor, double-click the new vault project name to make it the active project.
- 7 Enter the following information:
 - **Server:** The name of the computer housing the vault where this project is added. The name can be a computer name or an IP address.
 - **Database:** Identifies the database used by the vault project.
 - **Name:** The name of the vault where this project is added.
 - **Vault Project:** The name of this project preceded by \$/. The name should match the name you entered in the Project wizard.
 - **Virtual Folder:** Optional. The location of the folder used to publish data to share with customers. This location should be on a common server and mapped with a UNC path (for example, \\Server\\Share\\).


- Click Save to save the project file, and then click Cancel to close the dialog box.

NOTE Any settings except the Virtual Folder can be left blank and is entered automatically the first time you successfully log into the vault.

Create a single-user project

The single-user project is the default project. It is intended for designers whose files are not shared. All design files are in one workspace folder and its subdirectories, except for files referenced from libraries. The project file (.ipj) is stored in the workspace.



- 1 Click  ➤ Manage ➤ Projects.
- 2 In the Projects dialog box, click New to start the Project Wizard.
- 3 Select New Single User Project, and then click Next.
- 4 Name the project and specify the project location in the Project (Workspace) Folder box. Make sure it is a new folder that does not contain any data. The Project wizard creates a folder for the project file (.ipj) by default, but if you browse to a different location, the located folder name is used. Click Next.
- 5 Select existing libraries, add them to the project, and then click Finish.

NOTE In the Project wizard, Semi-isolated and Shared project types are unavailable by default. We recommend that you use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy projects types, click Tools tab ➤ Options panel ➤ Application Options ➤ General tab. Select the Enable creation of legacy project types check box.

If the check box is not selected, and Autodesk Vault is not installed, only a Single User project can be created with the Project Wizard.

Create legacy project types

Create a semi-isolated master project

We recommend that you use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy projects types, click Tools tab ➤

Options panel ► Application Options ► General tab. Select the Enable creation of legacy project types check box.

When working in a semi-isolated environment, you store shared files on the network in a workgroup. A master project specifies a shared workgroup. Each design team member creates a personal project (Semi-Isolated Workspace) to define where to create and edit files that are checked in to the shared workgroup. In each personal project, a path to the master project is included, automatically giving access to the shared locations and libraries.

- 1 Start Autodesk Inventor.
- 2 In the Projects dialog box, click New to start the Project wizard.
- 3 Select New Semi-Isolated Master Project, and then click Next.
- 4 Name the master project and specify the project location in the Project (Workgroup) Folder box. Make sure it is a new folder that does not contain any data. The Project wizard creates a folder for the project file (.ipj) by default, but if you browse to a different location, the located folder name is used. Click Next.
- 5 Select existing libraries and add them to the project, and then click Finish.

Create a semi-isolated workspace project

When working in a semi-isolated environment, shared files are stored on the network in a workgroup. A master project specifies a shared workgroup location. Each design team member creates a personal project (semi-isolated workspace) to define where to create and edit files that is checked in to the shared workgroup. In each personal project, a path to the master project is included, automatically giving access to the shared locations and libraries.

Designers check files in and out of the workgroup to edit in their workspace. When edited files are checked back in, other designers can refresh their files to see the changes.



- 1 Select ► Manage ► Projects.
- 2 In the Projects dialog box, click New to start the Project Wizard.
- 3 Select New Semi-Isolated Workspace Project, and then click Next.
- 4 Name the personal project and specify the project location in the Project (Workspace) Folder box. Choose a name that relates the personal project to the master project.


Make sure that the location is a new folder that does not contain any data. The Project wizard creates a folder for the project file (.ipj) by default, but if you browse to a different location, the located folder name is used. Click Next.

- 5 In the Master Project File box, browse to the appropriate master project. Only a project with a specified workgroup can be selected as a master project. The project path and name are shown in the Project Editor as the Included file.
- 6 Click Finish.

Create a shared project

When working in a shared environment, you store shared files on the network in a workgroup. All design team members use the same project and check files in and out of the workgroup for editing. No workspace is defined.



- 1 Select  > Manage > Projects.
- 2 In the Projects dialog box, click New to start the Project Wizard.
- 3 Select New Shared Project, and then click Next.
- 4 Name the project and specify the project location in the Project (Workgroup) Folder box. Make sure it is a new folder that does not contain any data. The Project wizard creates a folder for the project file (.ipj) by default, but if you browse to a different location, the located folder name is used. Click Next.
- 5 Select existing libraries and add them to the project, then click Finish.

After the project is created, use the Project Editor to customize options.

- 1 In the Project Editor, double-click the new project name to make it the active project.
- 2 For Use Styles Library, specify how a style library is used by the project. Choose one:
 - **Use Styles Library = Yes** uses the style library defined in the style library folder options. The library is writable so all designers can create and edit the styles and save them to the style library, replacing previous style definitions.

- **Use Style Library = Read Only** prohibits designers from saving new and changed styles to the style library. Library definitions cannot be replaced.
 - **Use Style Library = No** prevents a style library from being used. Styles used in project documents can be accessed only through styles created with the Style Editor, imported into the document, or present in the document template.
- 3 For Frequently Used Subfolders, right-click to add a shortcut to a subfolder of a project location from which you frequently open or save files.
 - 4 For Options, enter the following information:
 - **Old Versions to Keep on Save:** The default is one. -1 saves all versions.
 - **Using Unique Filenames:** Right-click and select Yes to indicate that no duplicate file names are used in the project or No if you use duplicate file names. If Yes, if the file is not found in the location stored in the reference, Autodesk Inventor searches the project folder structure for a unique file with a referenced file name. If No, the Resolve Files dialog box opens so you can locate the file manually.
 - **Name:** Right-click and select Rename to change the name of the project. The project name in the upper pane updates, but the shortcut name must be changed separately.
 - **Shortcut:** Right-click and select Edit or Delete. If you delete the shortcut, Autodesk Inventor recreates it automatically when you access the project.
 - **Owner:**The project owner, typically the lead engineer or CAD administrator. Optional.
 - **Release ID:**The version of the released project data. Useful to document a library folder. Optional.
 - 5 Click Save to save the project file, and then click Cancel to close the dialog box.

Edit projects

After you create a project, you can change some of its options, add or delete locations, or change its name. To reduce the possibility of file resolution problems, plan your project and folder structure for files associated with the project before you create it and start designing.

Keep in mind:


- If you attempt to edit a file that is checked out by someone else, a warning message alerts you to the file status.
- If you delete a location, Autodesk Inventor can no longer find referenced files from that location.
- Each project should have only one workgroup or workspace.

Before you edit a project, verify that all Autodesk Inventor files are closed. If you do not, the project is read-only.

TIP To see definitions of project options, click the Reference tab, and then choose Project Editor Reference.

Open the Project Editor and click a project to edit:



- 1 Within Autodesk Inventor, select  ➤ Manage ➤ Projects. Outside Autodesk Inventor, use the Microsoft Windows Start menu to select Programs ➤ Autodesk ➤ Autodesk Inventor [version] ➤ Tools ➤ Project Editor.
- 2 In the top pane, select the project to edit.
- 3 In the lower pane of the selected project, right-click the item to edit, and then select an option. Unavailable options are dimmed.
 - Change the project Type to Single, Shared, Semi-Isolated, or Vault.

NOTE The legacy project types, Semi-isolated and Shared project types are unavailable by default. We recommend that you use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy projects types, click Tools tab ➤ Options panel ➤ Application Options ➤ General tab. Select the Enable creation of legacy project types check box.

- For an Included file, choose Open, Edit, or Delete.
- For a Workspace, Workgroup, Frequently Used Subfolders, or Libraries, choose one:
For existing Workgroup search paths, select Edit, Add Path, Delete, or Move Up or Down.

Add path

Browse to the folder to add. Add a custom name, if appropriate.

Add paths from file	Browse to another project file. The paths from the file are added to the current project file.
Add paths from directory	Browse to any folder where you have files in one of its subfolders. A path for each subfolder of the folder you selected is added to the paths in the selected file location category.
Paste paths	Paste a path from the Clipboard into the selected section.
Delete section paths	Deletes all paths in the section.
Add proxy path	Adds a path to the library location for the folder that contains Mechanical Desktop part files.
Reset	Changes the location of Folder Options for Styles, Templates, and Content Center Files to the default specified on the Files tab of the Application Options dialog box.


- 4 Some Options entries may be edited. Right-click one or more:
 - For Options, choose Create Shortcut.
 - For Using Unique File Names, choose Yes or No.
 - For Name, choose Rename to give the project a new name.
 - For Shortcut, choose Edit or Delete.
- 5 Folder Options locations originate on the File tab of the Application Options dialog box. Right-click to edit individual locations.
- 6 Click Save, and then click Close.

Change the active project

The active project specifies the folders where you locate files specific to the project and the library files that those files reference. The active project also defines other options that are related to where styles files, template files, and other design data are located.

When you change projects, you change where Autodesk Inventor searches for referenced files. It also changes the file access dialogs, encouraging, and facilitating the recommended practice of only opening files from and saving files to locations in the active project.

Before changing the active project, close all files in Autodesk Inventor.

- 1 Select  ➤ Manage ➤ Projects.
- 2 In the Project dialog box, select a project from the list at the top of the dialog box.
- 3 Click Apply to make the selected project active.

NOTE If an existing project is not listed, click Browse to open the Choose Project File dialog box, and then search for it. The located project is added to the list.

Quick Reference

Projects

Activates and edits Autodesk Inventor projects. Accesses the Project wizard to create new project files.

To edit a project, first close all Autodesk Inventor files.

A project defines the locations of all files associated with a project, including where design data are stored, where you edit files, how many versions of the file are retained when you save a file, Content Center configuration settings, and the project type. Project file information is used to locate referenced files.

NOTE In the Project wizard, Semi-isolated and Shared project types are unavailable by default. We recommend that you use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy projects types, click Tools tab ➤ Options panel ➤ Application Options ➤ General tab. Select the Enable creation of legacy project types check box.

If Autodesk Vault is not installed and the Enable creation of legacy project types check box is not selected in Application Options, only a Single User project can be created with the Project wizard.

Technical note - project format change

The format of the project file changed in R10, but pre-R10 versions of Autodesk Inventor can read R10 and later project files. Because of the file format change, pre-R10 Autodesk Inventor cannot detect usable information in the project file and replaces the file.

Create a folder for R10 and later projects to isolate them from older project files. Set the Application Options ➤ File tab ➤ Projects Folder to a folder

that has R10 in its name and that contains only R10 and later projects. For example, name the folder "C:\My Documents\Autodesk Inventor\R10."

When you save a new project in R10 or later, a backup copy of the project file is placed in the Old Versions folder, so you can recover it if it is accidentally replaced. When you migrate a pre-R10 project file, and Autodesk Inventor detects a project file with the same name in the Old Versions folder, you are asked to restore the R10 or later project and use it instead of the older project.

Access:



In Autodesk Inventor, click ➤ Manage ➤ Projects.

In Design Assistant, click File ➤ Projects.

Outside Autodesk Inventor, on the Microsoft Windows Start menu, click Programs ➤ Autodesk ➤ Autodesk Inventor [version] ➤ Tools ➤ Project Editor.

Project Name/Project Locations Shows project file names and locations defined in the active project. When you pause the cursor over a path name, the tooltip shows the path. Click a path to display its subfolders and files in the main window.

Default Tutorial files and Samples install locations:

Microsoft Windows XP operating system

- Tutorial files: Program Files\Autodesk\Inventor [version]\Tutorial Files
- Sample files: Program Files\Autodesk\Inventor [version]\Samples

Microsoft Vista:

- Tutorial files: Users\login-name\AppData\Local\Autodesk\Inventor [version]\Tutorial Files
- Sample files: Users\Public\Documents\Autodesk\Inventor [version]\Samples

Select Project pane

The list of projects in the top pane of the Project Editor includes all projects that have a shortcut in the project folder. Three projects are installed with the Autodesk Inventor software: Default, samples, and tutorial_files. You cannot delete the Default project.

NOTE If a shortcut was deleted, its project is not listed. Click Browse to open the Choose Project File dialog box to search for it.

Click	Selects the project for editing and displays its paths and options in the bottom pane of the Project Editor. Selecting a project, does not automatically activate it.
Double-click	Selects and activates the project.
Right-click	<p>Displays options on the context menu for the selected project.</p> <p>Rename Opens the Name field where you can edit the name of a project that is not active.</p> <p>Browse Displays the Choose Project File dialog box so you can search for a project not listed. When you open a project that is not in the list, a shortcut is automatically created in the Projects folder.</p> <p>New Opens the Project Wizard so you can define a new project.</p> <p>Delete Deletes a project that is not active.</p>

Edit Project pane

The lower pane shows file locations, libraries, options, and settings that specify where files are stored, how many versions of the files to retain when you save, and the project type. Double-click any category to display its contents. To edit, right-click a category, and then click a menu option.

Type	<p>Identifies the type of the active project as single user, shared, semi-isolated, or vault. Activates the appropriate file check out warning system, and controls the file attributes of the workgroup for the project. The project type reverts to single user if you do not define a workgroup path. This option is set by the selected project type when you create a project. Select one of the following options:</p> <p>Single User: Use when you work alone and all the files in your project are in one location, except for library locations. Defines only a workspace, not a workgroup.</p> <p>Vault: When Autodesk Vault is installed, it isolates vaulted versions. Designers check out files to edit in a personal workspace and check back in to the vault. Other designers refresh files to see edits. Offers database queries, access to past configurations, and other utilities(only available if Autodesk Vault is installed). "Enable creation of legacy project types" check box must be selected in Application Options, to create a Semi-Isolated or Shared project.</p> <p>Semi-isolated: Uses a master project that defines shared locations (workgroup) and a personal project for each designer that includes the master project and specifies a personal workspace.</p> <p>Shared: Uses a shared workgroup location where all designers edit files. Use for small design teams with well-defined roles.</p>
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	<p>Single User: Use when you work alone and all the files in your project are in one location, except for library locations. Defines only a workspace, not a workgroup.</p>
Location	Shows the path to the project folder where the active project is stored.
Included File	Specifies the path to another project to include in the selected project. The search paths in the included file are included in the current file. The project type of the included file overrides the setting in the current file.
Use Style Library	<p>Specifies how a style library is used by the project.</p> <ul style="list-style-type: none"> ■ Use Style Library = Yes uses the style library defined in the style library folder options. The library is writable so all designers can create and edit the styles and save them to the style library, replacing previous style definitions. ■ Use Style Library = Read Only prohibits designers from saving new and changed styles to the style library. Library definitions cannot be replaced. ■ Use Style Library = No prevents a style library from being used. Styles used in project documents can be accessed only through styles created with the Style Editor, imported into the document, or present in the document template.
Workspace	<p>When the More button switches to an expanded project view, this section is visible. Shows the personal workspace specified when the project was created. Each project should have only one workspace where you edit and save files. Only vault, semi-isolated workspace, and single-user projects specify a workspace. By default, this section is hidden.</p>
Workgroup Search Path	<p>When the More button switches to an expanded project view, this section is visible. Shows the master workgroup for shared network locations where Autodesk Inventor searches for nonlibrary files. The workgroup is specified when the project is created. Each project should have only one workgroup. Design team members check files out from the workgroup to edit in a personal workspace. When finished, designers check changed files back in to the workgroup. Only semi-isolated master and shared projects specify a workgroup. By default, this section is hidden.</p>
Libraries	Shows paths to locations of files that are referenced and used, but not modified as part of the project. Each project can have one or more libraries.
Frequently Used Sub-folders	<p>Lists subfolders that are nested within a project workspace, workgroup, or library. Right-click to add a shortcut to a folder from which you frequently open or save</p>

files, or click the Plus command to browse to the location. The listed folders appear as shortcuts in the Open, Save, and Place dialog boxes, so you can quickly navigate to the location. They are not used to resolve file locations. Shortcuts to frequently used subfolders are preferred rather than adding nested workgroups or workspaces.

Folder Options

Identifies where project level files, such as templates and styles, are stored. Default locations are defined on the Files tab of the Application Options dialog box. Right-click individual locations to edit or select Reset to restore locations to default settings.

Design Data: Identifies where the project-specific style definitions are located.

Templates: Identifies where the new file templates for the project are located.

Content Center Files: Identifies the root folder for the Content Library files used by the current project.

Options

Shows global defaults for the selected project that were set when the project was created. The option settings in a project determine its file management functionality. Right-click an option to edit.

Old Versions to Keep On Save: Sets the number of versions to store in the *OldVersions* folder for each file saved. The first time a file is saved in a project, an *OldVersions* folder for that file is created. When the file is saved, the prior version is moved automatically to its *OldVersions* folder. After the number of old versions reaches the maximum, the oldest version is deleted when a newer version is moved into the folder. Click the variable, and then change it to the maximum number of file versions to keep in all *OldVersions* folders in the project.

Using Unique File Names: Specifies intent to create unique names for all files in the project (including the subfolder). Not applicable for library locations.

- Select Yes to indicate that no duplicate file names are used in the project. If a file is moved into the project, Autodesk Inventor searches through all editable project locations to find the file name, even if it was last accessed from a different folder.
- Select No to indicate that duplicate file names exist in the project. If No, and duplicate file names are found when resolving files, the Resolve Files dialog box opens so you can browse to the correct file to manually reestablish the link.

Name: Shows the name of the project. Right-click Name and select Edit to change the name. The name updates in the Select Project pane, but does not update the folder name.

Shortcut: Shows the name of the shortcut to the active project. Stores shortcuts in the projects folder. The project name is taken from the name of the target project file, not the name of the shortcut. Right-click the name of the shortcut,

and then click Edit. Enter a new name. You can select Delete to delete the shortcut. The shortcut is deleted from the projects folder, but the project is not deleted.

Owner: Identifies the project owner, typically the lead engineer or CAD administrator.

Release ID: Identifies the version of the released project data. If a project is used as a library by another project, the release ID may be useful in identifying which project to use.

Imported Components Folder Name: If you select to save imported component files to your Workspace, this folder is created and the component files are saved to it.

Imported Top Level Assemblies Folder Name: If you select to save imported assembly files to your Workspace, separate from your parts, this folder is created and the assembly files are saved to it.

Vault Options

Shows vault-related options only displayed for vault projects. Autodesk Vault must be installed to see these options.

Publish Folder: Specifies the location where Autodesk Vault publishes downloadable read-only versions of the files of the project.

Virtual Folder: Specifies the location where project data can be published to share with customers, collaborative design teams, and suppliers.

Right-click

Displays a context menu with options that are appropriate for the selected file path type.

Add Path: If a section is selected, adds a new path to the end of the section. If a path is selected, adds a new path after the selected path. You change the default name and path.

Add Paths from File: Opens the Choose Project File dialog box so you can select another project file. The paths from the file you select are added to the selected project file.

Add Paths from Directory: Opens the Browse for Folder dialog box so you can select a folder. A path for each subfolder of the folder you selected is added to the paths in the selected file location category.

Paste Path: Pastes a path from the Clipboard into the selected section.

Delete Section Paths: Deletes all existing paths from the selected section.

Commands

Editing commands

Add, edit, and change the order of search paths listed in the active project.



Moves a selected path up in the list of search paths.



Moves a selected path down in the list of search paths.



Adds a path to the selected set of search paths.



Sets the selected path to edit mode.



Expands and collapses the project view. Shows details of the location of the Workspace and Workgroups. When the recommended practice of only one editable location located at "." (usually a workspace) is used, this option is not used.



Find Duplicate Files: searches for duplicate file names in the editable locations of the current project. Duplicate files are listed in a dialog box, showing the file name, location, and relative path.



Edit Content Center Configuration: Displays the Configure Library dialog box so you can edit the configuration to specify which content is available.

Action commands

Initiate a global action on the entire project.

Autodesk Inventor Opens a session of Autodesk Inventor.

New Opens the Project wizard to create a project. The selected project paths become the default settings for the new project.

Browse Opens the Browse Project File dialog box to search for a project.

Save Saves the current edits to the selected project.

Done Closes the Project Editor. If the project is not saved, confirm the warning message.

Quick Reference

Project Wizard

The Project wizard creates a Autodesk Inventor project, sets the project type, name and location of the project file and the associated workgroup or workspace, and specifies the libraries to include in the project.

Technical note - project format change

The format of the project file changed in R10, but pre-R10 versions of Autodesk Inventor can read the R10 project file. Because of the file format change, pre-R10 Autodesk Inventor cannot detect usable information in the project file and replaces the file.

Create a folder for R10 projects to isolate them from older project files. Set the Application Options ► File tab ► Projects Folder to a folder that has R10 in its name and that contains only R10 projects. For example, name the folder "C:\My Documents\Autodesk Inventor\R10."

When you save a new project in R10, a backup copy of the project file is placed in the Old Versions folder, so you can recover it if it is accidentally replaced. When you migrate a pre-R10 project file, and Autodesk Inventor detects a project file with the same name in the Old Versions folder, restore the R10 project and use it instead of the older project.

Access:



In the active Autodesk Inventor window, click ► Manage ► Projects. In the project editor, click New.

Outside Autodesk Inventor, on the Microsoft Windows Start menu, click Programs ► Autodesk ► Autodesk Inventor [version] ► Tools ► Project Editor. In the Project Editor, click New.

NOTE In the Project Wizard, the legacy project types, Semi-isolated, and Shared project types are unavailable by default. We recommend that you use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy projects types, click Tools tab ► Options panel ► Application Options ► General tab. Select the Enable creation of legacy project types check box.

What type of project are you creating?

If Autodesk Vault is not installed (or you have not enabled Enable creation of legacy project types on the General tab in Application Options, only a Single User project can be created with the Project wizard.

Selects the project type and sets the corresponding project type. All projects use relative paths and place the project file (.ipj) at the root level (".\").

New Vault Project When Autodesk Vault is installed, creates a vault project and specifies one workspace, one or more libraries, and sets the project type to vault.

New Semi-Isolated Master Project Creates a master project that specifies one workgroup where files shared by the entire design team are stored, one or more libraries, and sets the project type to semi-isolated.

New Semi-Isolated Workspace Creates a personal project that includes the master project. The workgroup and library locations are inherited from the master project. Specifies one workspace and sets the project type to semi-isolated.

New Shared Project Creates a project shared by all design team members. Specifies one workgroup where shared files are located, one or more libraries, no workspace, and sets the project type to shared.

New Single User Project The default project type. Creates a project that specifies one workspace where data files are located, no workgroup, one or more libraries, and sets the project type to single user.

Project File

Specifies the project name, the location of the workgroup or workspace folder (according to project type), and the project file location.

Name Specifies a name and creates a project.

Project Folder Creates a workgroup (for shared files) or workspace (for personal edits) location, depending on the project type. Shows the location of the uniquely named new subfolder of the projects folder (where the .ipj is located). If you change the default name, the folder name also changes unless you browse to a specific folder. If so, Autodesk Inventor uses the located folder name and does not create a folder.

Project File to be created Shows the path to the named subfolder that represents the workgroup or workspace and is the location of the new project file (.ipj).

Master Project File When the project type is semi-isolated workspace, specifies the location of a project file to add in the Included file category in the new project. You can accept the path, edit the path, or click Browse to navigate to a different location.

Select Libraries

For vault, semi-isolated master, shared, and single-user projects, specifies existing libraries to include in the new project.

Semi-isolated workspace projects inherit libraries from the semi-isolated master project.

All Projects Lists all libraries in existing projects and their locations. To add libraries to the new project, select one or more libraries in the list, and then click Add.

New Project Lists libraries and their locations to include in the new project. To remove libraries, select one or more libraries in the list, and then click Remove.

Library Location Shows the path to the location of the selected library.

Quick Reference

Project Setup

Sets the project file for the active Design Assistant work session. Design Assistant uses the project file to determine the workspace and search paths available for the session.

Access: The Projects Setup dialog box is displayed if the default project location is not defined.

Project Folder:

Specifies the location of the Project Files to use for the active Design Assistant work session.

Project libraries

A library is a collection of related data files that you reference but usually don't modify, such as a released data set, a set of commonly used components, iPart factory parts, or standard parts. Libraries are located in a folder with a unique name separate from other data files. A single library can be used across any number of projects. You can use some or all files in a library.

In a project file, you right-click the Libraries location, and then use menu options to browse to a folder. By assigning the folder as a library, you indicate that the files are no longer editable using Autodesk Inventor. It is a good practice to use Microsoft Windows Explorer to set your library folders and files as read-only.

References to library files include the library name, as well as the relative path from the library folder to the referencing file. When resolving library references,

only the named library is searched. It is a good practice to avoid using duplicate file names whenever possible, even for parts in different folders or different libraries. If you cannot control where referenced parts originate or how they are named, files in different libraries might have the same name. Autodesk Inventor can resolve the reference because it knows which library to search in.

To share libraries across design workgroups

You can eliminate the need for each member of a design workgroup to set the names and paths manually in their individual projects for the libraries to share.

- 1 An administrator creates a common project with one or more library paths.
- 2 Each team member creates a personal project, using the common project as a template.

When you create a project using the project editor, select the common project before you click the New command. By default, the new project created by the wizard uses the selected project as a template, and has the same library definitions as the template project.

A project can include several kinds of libraries:

Standard content

Contains Autodesk Inventor parts or component libraries, such as:

- Autodesk Inventor Content Library.
- Third-party components.
- Company collections of commonly used components or other design elements, such as color and texture.

NOTE When you install Autodesk Inventor, you can set an option so that you can edit content library files. Use caution because other designs that reference the edited library file are affected.

iPart Factory

Contains Autodesk Inventor iParts that generate instances of parts, based on parameters you specify.

- The iPart Factory may be located in a library folder, and the parts you generate are kept in an associated proxy library. The two library folders bear the same name, except the name of the proxy library has a leading underscore (_) character.
- Do not make iPart proxy libraries read-only, because the factory has to create new files there.

Proxy libraries

The use of proxy libraries keeps the custom design files for a project separate from components that are purchased or re-used.

- There is only one line to change in the project file if you move a library, rather than update all the references in each individual file.
- File resolution is faster with the use of proxy libraries.
- Because libraries include the library name as part of the file name, file resolution is faster with library parts.

Another design set

This can be a whole or part of a design set that is a:

- Design that is already released and the parts should not be modified.
- An assembly or component you purchase from a third party, and cannot change.
- A project of another team that is in development. You can only reference it.

Collaborative design

Projects for collaborative design

In a collaborative design environment, several people can work on a project at the same time. An Autodesk Inventor project specifies the file locations for all the files and libraries in the project.

NOTE Autodesk Inventor warns you if you try to save a file that is not the latest version.

The project editor has several project types to support a collaborative design environment. When you create a project, you specify the project type, along with default locations for a workgroup or workspace. When a designer opens an Autodesk Inventor file, it contains references to other files.

NOTE In the Project Wizard, Semi-isolated and Shared project types are unavailable by default. We recommend that you use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy projects types, make changes on the General tab in Application Options. Select the Enable creation of legacy project types check box.

If the check box is not selected, and Autodesk Vault is not installed, only a Single User project can be created with the Project wizard.

Vault projects

Install Autodesk Vault to use Vault projects. The vault prevents designers from working on the "original" version of a file in the vault. Each designer creates a personal project that specifies a personal workspace and includes search paths to one or more master projects.

Files must be checked out of the vault before they can be edited. When edits are complete, they are checked back in, making the edits available to other designers on the team.

Using Autodesk Vault, you can set up queries regarding properties, references, past configurations, and other information.

To avoid file resolution problems, projects always use relative paths rather than absolute paths so that paths are relative to the project file location.

Single-user projects

The single user project is useful for individual designers. A personal workspace is defined where you create and edit files.

By default, all design files are saved in the workspace. You may specify one or more libraries.

You do not need to check files in and out for editing. The file status browser is not available because you are the only person with access to the files.

Shared projects

In shared projects, all files are stored on a server and are accessed by all designers. Shared projects are only appropriate for small design groups with well-defined tasks. A common project defines workgroup search paths and libraries.

Designers work on the "live" files rather than copying files to a personal workspace. File sharing requires everyone to update their assemblies to see the latest versions of files.

The shared project file is on the network and is used by everyone who works on the project. The project specifies one Workgroup search path and one or more libraries. For best results, no other type of search path is specified.

To avoid file resolution problems, projects always use relative paths rather than absolute paths so that paths are relative to the project file location (the workgroup, in shared projects).

When you open a file in Autodesk Inventor, you can use the file status browser to see the status of all saved files in the project and check files in and out.

Semi-isolated projects

A semi-isolated master project specifies the Workgroup for shared files and one or more libraries. Each designer also creates a personal project that includes the path to the master project (automatically allowing access to the shared workgroup and one or more libraries) and specifies a personal workspace for editing.

Designers check out files from the workgroup shared by the whole design team, which automatically copies the file into a personal workspace (specified in the personal project) for editing.

Components that are not in the personal workspace are referenced from the network locations. After you open a file in Autodesk Inventor, use the file status browser to see the status of all saved files in the project, and check files

in and out. Workgroup members do not see edits to files until they have been checked back in, and then they must refresh or reopen files to see the changes.

To avoid file resolution problems, projects always use relative paths rather than absolute paths so that paths are relative to the project file location (the workgroup for the master project; the workspace for personal projects).

Use Autodesk Inventor in workgroups

If you are working in a design team, the CAD Administrator typically sets up the working environment and develops the projects for team members. A project specifies the locations where Autodesk Inventor searches for files.

Before working on a collaborative project, specify the project to use and set options.

Set Autodesk Inventor project type

The project type is set when you create a project.

NOTE In the Project Wizard, Semi-isolated and Shared project types are unavailable by default. Use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy project types, click Tools tab ➤ Options panel ➤ Application Options ➤ General tab. Select the Enable creation of legacy project types check box.

If the check box is not selected, and Autodesk Vault is not installed, only a Single User project can be created with the Project Wizard.

Later, you can edit the project type but not all types are available.

- 1 In the Project Editor, lower pane, right-click Type.
- 2 Click the setting that is appropriate to your project: Off, Shared, Semi-Isolated, or Vault.
- 3 Click Save to save the new type.

Vault mode is only available after you install Autodesk Vault.

To check out files stored in a workgroup location

You check files out of a workgroup location, make changes, and then save them to your workspace location. When you check out a file from the workgroup location defined in a project, it is automatically copied to the workspace defined in that project.

Check out files from the file status browser in the Semi-Isolated and Shared project types.

- 1 In the assembly browser, right-click the title bar, and then click File Status on the menu. The File Status browser is displayed with status icons to the left of each file listed.
- 2 In the File Status browser, right-click a file name with the Available for checkout status icon in front of it.
- 3 Click Check Out on the menu.

To check in files to a workgroup location

You check files back in when you want to update the version in the workgroup location and make the changes available to others. For a file to check in successfully, its check-out properties must be the same in the workgroup location and the workspace.

- 1 In the assembly browser, right-click the title bar, and then click File Status on the menu. The File Status browser is displayed with status icons to the left of each file listed.
- 2 In the file status browser, right-click the file name with an icon in front of it indicating the file is checked out to you.
- 3 Click Check In on the menu. The file status browser icon changes to indicate that the file is no longer checked out to you.

To copy files to your workspace

When you check files out of the workgroup, the files are automatically copied to your workspace so you can edit them.

In some circumstances, you may need to copy a file from another location to work on it. You can use Design Assistant to copy the files or complete the following steps.

- 1 In Microsoft Windows Explorer, select the files, right-click, and then select Copy from the menu.
- 2 Navigate to your workspace folder, right-click, and then select Paste from the menu.

NOTE Always reserve the files before copying them to your workspace.

To update referenced files

As you work on a file in your workspace, its references to files on the network are maintained. Use the Refresh command to update your views.

Tools for working collaboratively

Autodesk Inventor provides tools for concurrent design so that members of a group can share design data and work in the context of the same assembly without making conflicting edits.

The tools include:

- A project editor that manages projects a designer is working on, defines the locations of all files associated with a project, provides quick navigation to key project locations, and simplifies moving and copying folders associated with a project without breaking cross file references.
- A file status browser in the shared and semi-isolated projects shows the status of all open files in a project, and checks files in and out to avoid conflicting edits.
- The semi-isolated project saves changes to files in a personal workspace. Changes are not visible to others on the design team until the edited files are checked in.
- Autodesk Vault, when installed, isolates source files from files checked out to designers. Changes must be checked in to the vault before they are visible to other designers. Database queries access file properties, references, and past configurations.
- The Engineer's Notebook captures design information and other notes.
- The Design Assistant tracks and manages file properties, links between files, and other important information about Autodesk Inventor files.

Additional tools are provided for extended design teams. Tools for extended design teams save all types of Autodesk Inventor design data into package formats. Package formats optimize both 2D and 3D digital CAD data for distributing outside the design department and viewing in Autodesk Streamline.

Once package files are published to Autodesk Streamline, all members of extended design teams can view, manipulate, and mark up files in a secure Web-based environment. It includes team members who may not own Autodesk Inventor.

See the Autodesk Streamline Web site for additional product information and purchase options.

File versions

Each time you save changes, a new version of the file is created in your workspace, and the previous version is moved to the *OldVersions* folder. Other designers who are referencing the file continue to see the earlier version of the file and do not see the new version until you check the file back in to the workgroup location in semi-isolated projects, or save the file in shared projects, and they refresh their view of the assembly.

What happens when you save a file?

Several actions happen automatically when you save a file.

- A new version is written to a new file, in the same folder.
- When the new file version is successfully on disk, the original file moves to a subfolder of the folder it was in named *OldVersions*, and is renamed to *file name.version.ext*, where version is a decimal version number. For example, *file name.0003.ext* would be the name for version 3 of *file name.ext*. If there are versions 2, 3, 4, 5 and 6 in the *OldVersions* folder, and you restore version 3 to be the current version, it is numbered version 7 to avoid conflicts with the existing versions 4, 5, and 6. Do not delete files from the *OldVersions* folder using Microsoft Windows Explorer. You could cause a problem in other Autodesk Inventor sessions.
- Finally, *file name.newVer.ext* is renamed to *file name.ext*.

Every save writes a new globally unique version ID to the file to identify it. Each Autodesk Inventor session remembers which version ID it is accessing. If you try to reopen the file at the normal location but the version ID has changed, Autodesk Inventor assumes someone else saved or checked it in, so the appropriate version of the file is opened from the *OldVersions* folder. The version of the file that is opened is used from that point forward to access additional information. Note the old version of the file is restored, but not the state of an assembly.

NOTE Files that were created before release 6 of Autodesk Inventor could have multiple versions in one file.

How do projects control file versions?

In a project, an option controls the number of versions saved in the OldVersions folders. In the lower pane of the project editor, click to expand Options. Click Old Versions To Keep On Save, and then right-click and select Edit from the pop-up context menu. Now enter a value in the field and press Enter to set the number of versions to keep.

After you save a file, other designers who reference the file continue to see the old version of the file until you check in the file to the workgroup.

If you work in a Shared or Semi-Isolated project, designers must use the Refresh command or reopen the assembly to update files that are referenced from the server.

Why would you use old versions of files?

Previous file versions can be restored when they are needed, but it is not possible to restore the previous state of an assembly.

The files in *OldVersions* are used for:

- A manual form of error recovery or backups.
- Opening earlier versions of files to load segments. If the earlier version of a file is not found, a message is displayed, and you Browse or Cancel:

Browse;

Browse to the *OldVersions* folder to find the file.

- **Open Old, No Save:** Open old version (Save not allowed): Open the old version. Keep in mind that you cannot save it.
- **Open Current:** Restore old version to current version: Restore the old version, and then open it. Not available when the current version is checked out to someone else.
- **Restore old, then Open:** Open current version: Open the current version. The open is redirected to the current version.

Cancel

Cancel the Save command.

How do file versions integrate with a PDM?

A facility provided through the PDM API allows some added control over the creation and management of *OldVersions* to accommodate the controls in PDM systems.

Set search paths

In a project file, search paths point to file locations where components associated with a project are stored.

When Autodesk Inventor searches for a file, it looks in the file locations in a project file. To avoid file resolution problems, projects always use relative paths rather than absolute paths so that paths are relative to the project file location.

- File names are stored in a relative path from the project root (".\"). If the file is in a library, the library name is also stored.
- A file name stored in the same folder or a subfolder of the file that references it is stored as a relative path.

When you create a project, a single workgroup or workspace is specified, depending on the project type. Some legacy projects may contain nested paths or multiple workgroups or workspaces. In the Project Editor, the nested paths are shown in red to remind you that they may introduce file resolution problems.

TIP Avoid nesting search paths. Define only one workgroup or workspace for a project.

Included file

You can use an Included file to add a path to a second project and therefore, gain access to its files. Any project type can use an Included file.

- You can use only one included file in the project.
- The project type in the included file overrides the type in the recipient project file.
- For a semi-isolated project, a master project can include the needed workgroup and library search paths and settings for the entire project. Each workgroup member can create a semi-isolated workspace project to set a personal workspace and specify only the Included file. Then workgroup members all have the same project settings and search paths from the Included file.
- If you work on a project with files in only a single project, do not use an included file.

Workspace

A personal workspace is created for vault, semi-isolated, and single-user modes. The workspace is intended for files that you edit and is accessible only by you. Changes you make to files in the workspace are not visible to other designers until you check them back in to a workgroup (for vault and semi-isolated modes). Other designers must refresh or reload the files to see the edits.

A workspace is not defined for shared mode.

- The workspace can contain only one location.
- Define only one workspace per project, in addition to any libraries you use.
- If the project type is shared, a workspace is not required.
- Do not set up library search paths as subfolders of the workspace, and do not create any other subfolders in the workspace folder.
- Maintain the same relative path to a file from the workgroup folder that would first be found when you copy the file to the workspace.
- Workspace is the first location searched for nonlibrary references.

Workgroup

A workgroup comprises the shared network locations where Autodesk Inventor searches for nonlibrary files. The workgroup designates the master storage location for data files for both shared and semi-isolated projects, but not single user projects.

- Set up only one workgroup per project. It is adequate for almost all situations. Do not specify its location as a subfolder of the workspace.
- You can organize your files into subfolders beneath the workgroup folder, and move the folder without breaking links to referenced files, if the path to the new location is specified in the project.
- If you plan to change the existing workgroup path, add the new one first before you delete the old workgroup path.
- Workgroup search paths are searched after the workspace is searched.

Libraries

A library is a storage location for files that are referenced.

- When you add a library search path, the folder in the path is considered a library.
- Any project type can include one or more libraries.
- A single library can be used in one or multiple projects.

Libraries for iParts require two folders: one for the iPart factory file and one for a corresponding proxy file that contains linking information. The folder names are the same except that the proxy folder has a leading underscore character (such as iParts and _iParts).

Tips for creating search paths

- To increase performance when you work with large assemblies, create libraries for read-only parts such as fasteners or purchased parts.
- Assign a meaningful unique name to each search path and library to avoid confusion when using File Open and Save and to maximize the usability of your datasets.
- To maximize portability of your datasets, define only one workspace or workgroup for the projects folder (which contains the .ipj file).


NOTE If using unique file names, set the project option Using Unique File Names to Yes. Autodesk Inventor automatically searches all locations in the project. If the file cannot be located, the Resolve Link dialog box opens and you locate the file manually.

Set search paths for projects

You use the Project Editor to set search paths to file locations where you access and edit files.

To access the Project Editor:

- 1 Choose either method:

- In Autodesk Inventor, select  ➤ Manage ➤ Projects.
- Outside Autodesk Inventor, select the Start menu ➤ Programs ➤ Autodesk ➤ Autodesk Inventor [version] ➤ Tools ➤ Project Editor.

- 2 In the Project Editor, select the project to edit in the upper pane.
- 3 In the lower pane, right-click a path category, and then select an option on the menu.

Set the path for an included file

You can include the search path to another project file in the selected project. An included project is often a master workgroup project located on the network. Using this method, you can access all file locations from the included file.

- 1 Right-click Included File, and then click Edit on the menu.
- 2 Browse to the project file you want to include and select it.
- 3 Click Save.

Set a path for a workspace

A workspace is intended to contain files you create and edit and is not accessible by other designers.

Use only one workspace for each project. In most cases, the workspace is already created for you by the Project Wizard when you create a vault, semi-isolated workspace, or single-user project. The default location is at the root level (where the project file is located).

- 1 Right-click Workspace, and then click Add Path on the menu.
- 2 Accept the path shown, edit the path, or browse to and select a workspace folder.
- 3 Click Save.

NOTE If you work in a Single User or Semi-isolated Workspace project, you can copy files to your workspace to work on them. You can use Design Assistant to copy the files, or when you check them out of a workgroup location, they are automatically copied to your workspace. In the File Status browser, click a name with an icon to the left indicating it is available for checkout, and then click Check Out.

Set workgroup search paths

A workgroup is shared network locations that designates the master storage for data files for both shared and semi-isolated projects.

Each project has only one workgroup. When you create a shared or semi-isolated master project, a workgroup is automatically designated.

You may need to change the workgroup location. Be sure to add the new workgroup location before you delete the old one.

- 1 Right-click Workgroup Search Paths, and then click one of the options on the menu:

Add Path	Enter a library name and location in the fields provided, and then click Save.
Add Paths from File	Browse to and select a project (.ipj) file, and then click Open.
Add Path from Directory	Browse to and select a folder that contains workgroup files. A path for each subfolder of the folder you selected is added to the paths in the selected file location category."
Paste Path	Paste a copied path into the field provided.
Delete Section Path	Deletes all paths in the Workgroup section. If you delete all paths, the project type is automatically reset to Single User.

- 2 Click Save.

Set library locations

You can specify paths to standard and custom libraries of read-only files. Each project can have multiple libraries. When you specify a location as a library, the files cannot be edited, but can be referenced.

- 1 Right-click Libraries and click one of the options on the menu:

Add Path	Enter a library name and its location in the fields provided, and then click Save.
Add Paths from File	Browse to and select a project (.ipj) file, and then click Open
Paste Path	Paste a copied path into the field provided.
Delete Section Paths	Delete all paths in the Library section.

- 2 Click Save.

NOTE When you create a project, you can add library search paths from other projects.

- 1 Verify that all Autodesk Inventor files are closed.
- 2 Open the Project Editor, and then click New.
- 3 On the first two pages of the Project wizard, specify the project type, the project name, and location of the workspace or workgroup. Click Next.
- 4 On the Select Libraries page, use the Add and Remove arrows to create a list in the New Project pane. Click Finish.

NOTE When Autodesk Inventor is installed, a setting can be selected that allows Content Library files to be edited. You must reinstall Autodesk Inventor to reset this option.

Find where files are used

In Design Assistant, you can search for all files that reference a specified file. For example, if a part file is used in multiple assemblies, you can view a list of all assemblies.

Where Used search returns only first level references. For example, if you create a drawing of an assembly, the search recognizes the assembly as referenced by the drawing but does not recognize the components as referenced by the drawing.



- 1 Open Design Assistant, either within Autodesk Inventor or from Microsoft Windows Explorer.
- 2 Select Tools ► Find ► Where Used to open the dialog box.
- 3 Specify the file that is the subject of the search.
- 4 Specify the search paths for the files to find. If necessary, you can specify multiple search paths.
- 5 Specify what file types you are searching for by checking the filter options.
- 6 Click Search Now to initiate the search. You can save the search results to a text file and view them using a text editor.

Find specific objects in Autodesk Inventor files

You can search for specific objects, such as constraints, components, features, sketches, and welds in all Autodesk Inventor file types. You can refine your searches by including additional criteria such as and/or operators, properties, conditions, and values. You can specify the location in which you want to search.

Find is useful when you want to locate sick constraints, suppressed components or features, sketch parameter names or values, or references, as well as any other combination of criteria you select.

Find in Window

Use the Find in Window shortcut to locate an object in the graphics window quickly.

- 1 In the browser, expand folders to locate the object you want to find.
- 2 Select one or multiple components, constraints, or other objects.
- 3 Select one or multiple constraints or other objects.
- 4 Right-click and select Find in Window. The found objects are zoomed and centered in the graphics window.

How to access the Find dialog box

To search using specific criteria, open the Find dialog box to initiate a search in any of the following ways:

Design Assistant	Select Tools ► Find ► Autodesk Inventor Files from the menu.
Autodesk Inventor Open dialog box	Click Find.
Autodesk Inventor ribbon	When a file is active, select Tools tab ► Find panel ► Find Component or click the binoculars icon on the browser menu bar.
Windows Start menu	Select Search ► Autodesk Inventor Files.

- 1 In the Find dialog box, set the search criteria. The criteria may be as simple as files of a given type or may include specific values for one or more file properties.
If desired, click Open Search to search on previously specified criteria.
- 2 Select the object type from the Look For list.
- 3 In the Define more criteria box, you can make your search more specific. If desired, enter one or more:
 - Using the And/Or operators, select And to accumulate criteria or select Or to satisfy the search if either criteria is met.

- Specify Property. The Find command recognizes the active environment and offers search criteria based on the environment.
- Select a Condition.
- Specify a Value to complete the Property, Condition, Value statement.

For example, specify a Stock Number (property) that is exactly (condition) A97000_3_8 (value). Or, specify Suppress status (property) equals (conditions) Suppressed (value).

- 4 Click Add to List to add the refined criteria to the search box.
- 5 Specify the current path file.
- 6 Click Find Now to initiate the search. You can save the search results to a text file and view them using a text editor.

Quick Reference

Where Used

Searches for the files in the active project that directly reference the specified file. You can search for references to any Autodesk Inventor file as well as to word processing documents, spreadsheets, and other valid files that are referenced in an Autodesk Inventor file.

Access:



Select Tools ► Find ► Where Used from Design Assistant menu.

Search for references to file	Specifies the file that is the subject of the search. Enter the file name in the box or click Browse to find the file. If you enter the path to the file, the search returns only references that match the path. To search for a reference to the file name regardless of where it is found, enter only the file name.
Look in	Sets the paths to search for the files. To add a path to the list, click the Click to Add prompt and select a folder to include its path in the list. To remove a path from the list, select it and press the Delete key.
File types	Filters the search results to the file types specified. At least one file type must be selected before initiating the search.

Include subfolders	Includes the subfolders in the specified search paths when searching for related documents. Select the check box to search the specified search paths and all subfolders. Clear the check box to search only in the specified search paths.
Search Now	Searches the specified folders for files that reference the selected file, and expands the dialog box to display the list of files found.
New Search	Clears the Search for References to File box so that you can set up a new search.
Close	Closes the dialog box.
Files Found	Displays the list of files in the specified search paths that reference the specified file. If you opened Design Assistant from Microsoft Windows Explorer, you can right-click any file in the list and select Design Assistant to open a Design Assistant session for the selected file.
Stop Search	Interrupts the active search operation.
Save List	Opens the Save As dialog box so that you can save the Files Found list as a text file.

Quick Reference

Find

Defines searches for constraints, components, features, sketches, and welds. When searching for any object, all the browser nodes that contain searched objects are expanded to make selections more visible.

NOTE If Assembly view is active in the browser, each constraint half is listed under the feature or component it constrains. Find locates the first occurrence in the browser. If the second half of the constraint is the desired search target, use the context menu option Other Half.

Access:

From Design Assistant, select Tools ➤ Find ➤ Autodesk Inventor Files from the menu.
 From the Autodesk Inventor Open dialog box, click Find.
 From the Autodesk Inventor ribbon when a file is active, select Tools tab ➤ Find panel ➤ Find Component or click the binoculars icon on the browser menu bar.
 From the Windows Start menu, select Search ➤ For Files or Folders ➤ Autodesk Inventor Files.

Find files that match these criteria

Sets the conditions for the search.

Search criteria	Lists the active search criteria.					
Look For	Defines the type of object to search for.					
Open Search	Opens a list of previously saved searches so that you can use their criteria for the current search.					
Save Search	Opens the Save Search dialog box so that you can save the current search to use it again. Enter a unique name for the search.					
Match Case	When selected, causes the search to observe the capitalization you use in the search criteria.					
Delete Line	Removes the selected search criterion from the list.					
Clear Search	Removes all items from the list of search criteria.					
Define More Criteria	<p>Sets up search criteria that you can add to the current search.</p> <p>And/Or sets the relationship of the new criterion to the other criteria in the list. Select And to find only files that contain this criterion in addition to previously defined criteria. Select Or to find files that contain any of the specified criteria.</p> <p>Property selects a file property as the criterion. The list of properties is adjusted to the selected object type. The Find command recognizes which environment is active and adjusts the listed properties accordingly. For example:</p> <table><tr><td>Assembly and Weldment environment</td><td>Components, constraints, features, sketches, and welds</td></tr><tr><td>Part environment</td><td>Features and sketches</td></tr></table> <p>File: Open dialog box, Design Assistant, or Windows Search</p> <p>File types (.ipt, .iam, .idw, .dwg (Autodesk Inventor Drawing), .ide, .ipn)</p> <p>Click the arrow and select from the list. To specify a custom property, enter the property name in the box.</p> <p>Condition sets the condition for the specified property value. Click the arrow and select from the list.</p> <p>Value specifies the value of the selected property. After setting the property and condition, enter the value in the box.</p>		Assembly and Weldment environment	Components, constraints, features, sketches, and welds	Part environment	Features and sketches
Assembly and Weldment environment	Components, constraints, features, sketches, and welds					
Part environment	Features and sketches					

Add To List adds the defined criterion to the list of search criteria.

Search Location

Specifies where to search for the files.

Current Path File Limits the search to the files in the specified folder.

Look in Sets the folder or file to search. Click Browse to find the desired search location.

Find Now

Searches the target location for files that meet the specified criteria. The list of files is displayed in the Files Found dialog box. To save the list to a text file, click the Save List command in the Files Found dialog box and specify the file name.

Quick Reference

Find String

Specifies a string (such as a referenced file name) and searches for it in all files listed in the Design Assistant Manager browser.

Access:

Use Microsoft Windows Explorer to select an assembly file, right-click, and then select Design Assistant.

Click the Manage icon to activate the Design Assistant Manager.

Select Tools ► Find ► String from the menu.

Search String

Enter the string into the box, and then click OK to initiate the search. When the search is complete, the file names are highlighted in the Design Assistant Manager browser.

NOTE The search on the string is case sensitive.

Paths for iPart factory parts

In a project file, search paths point to file locations where components associated with a project are stored.

When Autodesk Inventor searches for a file, it looks in the file locations in a project file. To avoid file resolution problems, projects always use relative paths rather than absolute paths so that paths are relative to the project file location.

- File names are stored in a relative path from the project root (".\"). If the file is in a library, the library name is also stored.
- A file name stored in the same folder or a subfolder of the file that references it is stored as a relative path.

When you create a project, a single workgroup or workspace is specified, depending on the project type. Some legacy projects may contain nested paths or multiple workgroups or workspaces. In the Project Editor, the nested paths are shown in red to remind you that they may introduce file resolution problems.

TIP Avoid nesting search paths. Define only one workgroup or workspace for a project.

Included file

You can use an Included file to add a path to a second project and therefore, gain access to its files. Any project type can use an Included file.

- You can use only one included file in the project.
- The project type in the included file overrides the type in the recipient project file.
- For a semi-isolated project, a master project can include the needed workgroup and library search paths and settings for the entire project. Each workgroup member can create a semi-isolated workspace project to set a personal workspace and specify only the Included file. Then workgroup members all have the same project settings and search paths from the Included file.
- If you work on a project with files in only a single project, do not use an included file.

Workspace

A personal workspace is created for vault, semi-isolated, and single-user modes. The workspace is intended for files that you edit and is accessible only by you. Changes you make to files in the workspace are not visible to other designers

until you check them back in to a workgroup (for vault and semi-isolated modes). Other designers must refresh or reload the files to see the edits.

A workspace is not defined for shared mode.

- The workspace can contain only one location.
- Define only one workspace per project, in addition to any libraries you use.
- If the project type is shared, a workspace is not required.
- Do not set up library search paths as subfolders of the workspace, and do not create any other subfolders in the workspace folder.
- Maintain the same relative path to a file from the workgroup folder that would first be found when you copy the file to the workspace.
- Workspace is the first location searched for nonlibrary references.

Workgroup

A workgroup comprises the shared network locations where Autodesk Inventor searches for nonlibrary files. The workgroup designates the master storage location for data files for both shared and semi-isolated projects, but not single user projects.

- Set up only one workgroup per project. It is adequate for almost all situations. Do not specify its location as a subfolder of the workspace.
- You can organize your files into subfolders beneath the workgroup folder, and move the folder without breaking links to referenced files, if the path to the new location is specified in the project.
- If you plan to change the existing workgroup path, add the new one first before you delete the old workgroup path.
- Workgroup search paths are searched after the workspace is searched.

Libraries

A library is a storage location for files that are referenced.

- When you add a library search path, the folder in the path is considered a library.
- Any project type can include one or more libraries.

- A single library can be used in one or multiple projects.

Libraries for iParts require two folders: one for the iPart factory file and one for a corresponding proxy file that contains linking information. The folder names are the same except that the proxy folder has a leading underscore character (such as iParts and _iParts).

Tips for creating search paths

- To increase performance when you work with large assemblies, create libraries for read-only parts such as fasteners or purchased parts.
- Assign a meaningful unique name to each search path and library to avoid confusion when using File Open and Save and to maximize the usability of your datasets.
- To maximize portability of your datasets, define only one workspace or workgroup for the projects folder (which contains the .ipj file).

NOTE If using unique file names, set the project option Using Unique File Names to Yes. Autodesk Inventor automatically searches all locations in the project. If the file cannot be located, the Resolve Link dialog box opens and you locate the file manually.

Set paths for iPart factories in projects

In a project file, define a library for both the folder containing the iPart factory part files and a corresponding proxy folder. After the libraries are added to the project, you can add the iParts to an assembly.

- 1 Verify that all Autodesk Inventor files are closed.
- 2 In the Project Editor, double-click a project to activate it.
- 3 In the lower pane, right-click the Libraries category, and then click Add Path. Browse to the folder or enter the name of the library folder to add a new library path for the iPart factory file.
- 4 If the folder was created previously, right-click the newly created library path, and then click Add Proxy Path on the menu. Browse to the folder location and select it.

For custom iPart factories only, enter a location for the iPart factory proxy file. A proxy folder is created automatically with the same name as the iPart factory library, but with a leading underscore (_) character.

If the proxy folder does not exist, use Microsoft Windows Explorer to create it before you add the proxy path or use Add Path again and specify the new folder path. Name the new library the same as the iPart factory library preceded with an underscore character.

Create read-only templates for a shared server

A common template eliminates the most repetitive tasks. Templates are easy to construct and provide significant time saving benefits. You can create templates with different units of measurement, drafting standards, file properties, or other defaults, and then use them to create new files. Save your customized templates with a meaningful name. Once you save the file in the Templates folder it displays in the New File dialog box. Do not overwrite the default templates: you may need them again.

Part and assembly templates set the default units of measurement, file properties, and other defaults for models. Drawing templates set the drafting standard, drawing resources, file properties, and other defaults for drawings. Presentation templates set the file properties for assembly presentations.

For detailed information on creating template files to increase productivity among your designers see the Autodesk Inventor Help.

Part Templates:

Consider the following:

Turn on origin planes: When creating part models, it is a good practice to reference work planes and origin planes as much as possible. To make it easier, turn on the origin planes. Make all three origin planes visible for easy reference.

Set Dimension Display: Tailor dimension display conditions for your requirements.

Set Default Tolerances: Customizing default tolerances for dimension values helps you match part feature and sketch dimensions to the standards of your company with appropriate precision.

Set the default Material and Color Style: To make your mass properties more accurate, set the default parts material in the Styles and Standards Editor.

Create Custom Properties: Custom properties are useful for adding information to bills of materials, parts lists, and other drawing annotations.

- 1 In Autodesk Inventor, select the Help button.
- 2 Select the Contents tab, and navigate to the Help topics:

- 3 Autodesk Inventor ► Parts ► Part modeling overview ► [Part templates](#) on page 122

Assembly templates

Consider the following:

- Change the sizes of the origin work planes to better accommodate the average size of the assemblies you create.
- If you use differing units of measurement in different assemblies, create different templates for each.
- Define the columns and format for the bill of materials.

- 1 In Autodesk Inventor, click Help.
- 2 Select the Contents tab, and navigate to the Help topics:
- 3 Autodesk Inventor ► Assemblies ► Build assemblies ► Fundamentals: Templates for assemblies

Sheet metal

Consider the following:

- Any model geometry that is useful as a starting point including faces or contour flange base features. Sheet metal styles for bends, sheets, corners, and material type. These values are the default for all sheet metal parts based on the template.
- Parameter names and values you customize in the Parameters dialog box. Establish your own parameter names, enter instructions or other explanatory text in the parameters list, or set up your own parameters.
- Change the sizes of the origin work planes to better accommodate the average size of the sheet metal parts you create.
- If you use specific settings for certain types of sheet metal parts, create different templates for each using the appropriate sheet metal style.

- 1 In Autodesk Inventor, click Help.
- 2 Select the Contents tab, and navigate to the Help topics:
- 3 Autodesk Inventor ► Parts ► Sheet Metal Parts ► Templates for Sheet Metal parts: [Sheet metal templates](#) on page 125

Drawing templates

There are some components it is useful to define in your customized drawing template. For detailed information refer to the following topics in the online Help:

- 1 In Autodesk Inventor, click Help.
- 2 Select the Contents tab, and navigate to the Help topics:
- 3 Autodesk Inventor ► Drawings ► Drawings overview ► Templates for drawings: [Create drawing templates](#) on page 129

IMPORTANT Remember to click **Back** to return to the CAD Manager Guide if you click any of the links on this page.

Part templates

When you install Autodesk Inventor, your selection of default units of measurement sets the default template used to create parts. Use this template or another predefined template, modify one of the predefined templates, or create your own template.

A part file becomes a template when you save it in the Templates folder. For example, if you have a part file that contains settings, properties, or other elements you want to use for other parts, save a copy of it in the Autodesk\Inventor [version]\Templates folder. The next time you create a part file, the new template is available.

The location of the Templates folder is by default:

Microsoft Windows XP: Program Files\Autodesk\Inventor [version]\Templates.

Microsoft Vista: Users\Public\Documents\Autodesk\Inventor [version]\Templates.

Deciding what to include in a part template

Any settings that streamline the creation of parts or establish default values can be included in a template.

- Change the grid settings and the size of the origin work planes to accommodate the average sizes of the parts you create.
- Create a default sketch that has default work features, such as a work axis at the intersection of the three base work planes.

- Create a sketch on a plane other than XY to change the definition of the Home View.
- Add sketches or solid geometry as the start for commonly created parts.
- If you use different units of measurement in different parts, create different templates for each.
- Define commonly used parameters or link to a spreadsheet of parameters.
- Set and define the default material.
- If you use lighting and color styles, set the styles so that they are available when you want them.

Setting properties

You can specify properties such as cost center, project name, or manager and save them as part of the template. Properties can be used to find, track, and manage your files. They can also be used to add and maintain information automatically in title blocks, parts lists, and bills of material.

Setting a default part template

You can set any template as the default template. Save the file in the Templates folder with the name *standard.ipt*. To avoid overwriting the existing default template, move or rename it before saving the new template.

Files that reside in the Templates folder appear on the Default tab of the New dialog box when you create new files. Files that reside in a subfolder of the Templates folder appear on other tabs in the New dialog box.

NOTE To add tabs to the New dialog box, create new subfolders in the Templates folder, and add template files to them. The New dialog displays a tab for each subfolder in the Templates folder.

Assembly templates

Each new assembly is created from a template. When you install Autodesk Inventor, your selection of default units of measurement sets the default template used to create assemblies. You can use this template or another predefined template, modify one of the predefined templates, or create your own templates.

Any assembly file can be used as a template when you save it in the Templates folder. For example, if you have an assembly file that contains settings and properties you want to use for other assemblies, save a copy of it in the Autodesk\Inventor [version]\Templates folder. The next time you create an assembly file, the new template is available.

The location of the Templates folder is by default:

Microsoft Windows XP: Program Files\Autodesk\Inventor [version]\Templates.

Microsoft Vista: Users\Public\Documents\Autodesk\Inventor [version]\Templates.

Deciding what to include in an assembly template

Any settings that streamline the creation of assemblies or established default values can be included in a template.

- Change the sizes of the origin work planes to accommodate the average size of the assemblies you create.
- If you use differing units of measurement in different assemblies, create different templates for each.
- If you use lighting and color styles, set the styles so that they are available when you want them. Associate the template file with a style library, or if you do not use style libraries, define the styles in the template file.
- Define the columns and format for the bill of materials.

Setting properties

You can specify properties such as cost center, project name, or manager and save them as part of the template. Use Properties to find, track, and manage files. They can also be used to add and maintain information automatically in title blocks, parts lists, and bills of material.

Setting a default assembly template

You can set any template to be the default template for creating new assemblies. To make a template the default, save it in the Templates folder with the file name *standard.iam*. To avoid overwriting the existing default template move or rename the existing standard template before saving the new template.

Files that reside in the Templates folder appear on the Default tab of the New dialog box when you create new files. Files that reside in a subfolder of the Templates folder appear on other tabs in the New dialog box.

NOTE To add tabs to the New dialog box, create new subfolders in the Templates folder and add template files to them. The New dialog box displays a tab for each subfolder in the Templates folder.

Sheet metal templates

Each new sheet metal part is created from a template. When you use the default sheet metal template installed with Autodesk Inventor, the default material style, sheet metal rule and unfold rules are applied to the sheet metal parts created. You can use this template or another predefined sheet metal template (English or Metric for example), modify one of the predefined templates, or create your own templates by copying and modifying one of the existing templates.

TIP If you routinely create sheet metal parts using a specific material or thickness, or parts which are produced using specific machines or tooling within your shop, consider creating unique templates for these commonly created model types.

Any sheet metal file can be used as a template when you save it in the Templates folder. For example, if you have a sheet metal file that contains settings and properties you want to use for other sheet metal parts, save a copy of it in the Autodesk\Inventor [version]\Templates folder. The next time you create a sheet metal file, the new template is available.

Deciding what to include in a sheet metal template

You can define a style in a sheet metal file and save it as a template, and then use it as the basis for new sheet metal parts. Include the following information:


- Any model geometry that is useful as a starting point including faces or contour flange base features.
- Sheet metal styles for bends, sheets, corners, and material type. These values are the default for all sheet metal parts based on the template.
- Parameter names and values you customize in the Parameters dialog box. Establish your own parameter names, enter instructions or other explanatory text in the parameters list, or set up your own parameters.

- Change the sizes of the origin work planes to accommodate the average size of the sheet metal parts you create.
- If you use specific settings for certain types of sheet metal parts, create different templates for each using the appropriate sheet metal styles.

Setting iProperties

Specify iProperties such as cost center, project name, or manager and save them as part of the template. Use iProperties to find, track, and manage files, and automatically add and maintain information in title blocks, parts lists, and bills of material.



Click  iProperties to open the iProperties dialog box, and then edit the appropriate properties on the various dialog box tabs as required.

Setting a default sheet metal template

Set any template to be the default template for creating new sheet metal parts. To make a template the default, save it in the Templates folder with the file name *sheet metal.ipt*. To avoid overwriting the existing default template, move or rename it before saving the new template.

Files that reside in the Templates folder appear on the Default tab of the New dialog box when you create new files. Files that reside in a subfolder of the Templates folder appear on other tabs in the New dialog box.

NOTE To add tabs to the New dialog box, create new subfolders in the Templates folder and add template files to them. The New dialog box displays a tab for each subfolder in the Templates folder.

Drawing templates

Each new drawing is created from a template. When you install Autodesk Inventor, your selection of a default drafting standard sets the default template used to create drawings. Use this template or another predefined template, modify one of the predefined templates, or create your own templates to enforce conventions.

Create a template from a drawing, preserving annotations on drawing sheets, such as custom symbols, notes, and revision tables. Your borders, title blocks, and view definitions can be retained in the template. View annotations and general notes are not saved in a template.

A drawing becomes a template when you save it in the Templates folder. For example, if you have a drawing file that contains the setup you want to use for other drawings, save a copy of it in the Autodesk\Inventor [version]\Templates folder. The next time you create a drawing file, the new template is available.

The location of the Templates folder is by default:

Microsoft Windows XP: Program Files\Autodesk\Inventor [version]\Templates.

Microsoft Vista: Users\Public\Documents\Autodesk\Inventor [version]\Templates.

When you create a drawing, it is automatically assigned an active drafting standard that controls the styles used to format dimensions, text, line weights, terminators, and other elements that are dictated. You can use the default standard or select from another named standard (ANSI, BSI, DIN, GB, ISO, or JIS).

The active standard has a default set of styles. Add or edit styles in the current document, and if you want other designers to use the custom styles, save them to the style library. Usually, the style library, which contains a master version of all styles, is managed by the CAD Administrator.

Autodesk Inventor drawings also include elements that are not part of the named standard. Copy, edit, or delete the elements as needed to customize them for your company. Save the changes to sheets and views, for example, in the drawing template, so they are available to other designers.

If you must supply customers with AutoCAD files, create a DWG template in Autodesk Inventor using an AutoCAD file that contains the appropriate layers, blocks, title blocks, and borders you need to deliver to your AutoCAD customers. Specify the object defaults for these properties in the Styles Editor.

NOTE Autodesk Inventor needs a valid drawing template file when opening AutoCAD .dwg files directly. By default, Autodesk Inventor uses the Standard drawing template files (Standard.idw or Standard.dwg) located in the Templates folder for the current project or in the Default Templates location. Override this behavior by specifying a full path and file name to any Autodesk Inventor drawing (*.idw;*.dwg) in the following registry key:

KEY_CURRENT_USER\Software\Autodesk\Inventor\Registry\Version [version]\AppData\DrawingLayout\Preferences\General\Standard Template Override

Setting the drafting standard

To ensure the correct drafting standard in all drawings, specify the drafting standard in the template, and then use the Style and Standard Editor to customize its styles to set formatting attributes.

If you use different drafting standards at different times, you can create one template for each standard. Each drawing has only one active standard and it specifies the available styles.

Setting up drawing resources

The Drawing Resources folder in the Drawing browser contains folders for sheet formats, title blocks, borders, and sketched symbols that you can use to add and set up new sheets. You can customize or add to the drawing resources, and then save them in your template file.

Tips:

- Create title block formats, custom borders, and sketched symbols before creating the sheet formats.
- Copy and paste resources from the browser of one drawing to the browser of another.
- Rename sheet formats, borders, title blocks, or sketched symbols. Select the object to rename and slow-click the name to enclose it in an edit box, and then enter a new name.
- Sort the drawing resources by name in the browser. Right-click the Sheet Formats, Borders, Title Blocks, or Sketched Symbols entry, and then select Sort by Name.
- Reorder the drawing resources. Drag a sheet format, border, title block, or sketched symbol and drop it at the appropriate position in the browser.
- To copy drawing resources from one source file to multiple destination files, use the Drawing Resource Transfer wizard.

Setting up sheets in a template

After defining sheets in drawing resources, you can add them to a template. Add a sheet format to the template for each sheet definition you need in a new drawing.

The sheets in a drawing can each be created with a different sheet format. To change the format of the first sheet in the drawing, add a sheet with the appropriate format, and then delete the first sheet.

Adding default views

You can add default base views or projected views to the sheet formats in a template. Use any file to add the views. The template saves the information about the standard views. When you create a file with the template, you are prompted to select the model file from which to create the views.

Setting properties

Specify properties such as cost center, project name, or manager, and then save them as part of the template. Use Properties to add and maintain information automatically in title blocks, borders, sketched symbols, and text, or to track and manage files using Design Assistant.

Setting a default drawing template

Any template can be the default template for new drawings. To make a template the default, save it in the Templates folder with the file name *standard.idw* or *standard.dwg*. To avoid overwriting the existing default template, move or rename the existing standard template before saving the new template.

Files that reside in the Templates folder appear on the Default tab of the New dialog box when you create new files. Files that reside in a subfolder of the Templates folder appear on other tabs in the New dialog box.

NOTE To add tabs to the New dialog box, create new subfolders in the Templates folder and add template files to them. The New dialog displays a tab for each subfolder in the Templates folder.

Create drawing templates

Create drawing templates

All new drawing files are created with a template. A default template is provided for each drafting standard or you can create your own templates.

You can create a template from a drawing, preserving annotations on drawing sheets, such as custom symbols, notes, and revision tables. In addition, your borders, title blocks and view definitions can be retained in the template. View annotations and general notes are not saved in a template.

NOTE The default drawing templates are named *standard.[file extension]* and are located in the Templates folder. To replace the default template, rename *standard.[file extension]* to avoid deleting it, and then replace it with the customized template named *standard.[file extension]*.

To replace or create a IDW default template



- 1 Create a drawing using an existing template based on the appropriate drafting standard (such as ANSI, ISO, or DIN).
- 2 Customize as needed:
 - Modify, create, or import title blocks and borders to conform to your requirements.
 - Create custom symbols for reusable standardized content.
 - Review style definitions and settings contained in the active, default standard style and modify or add settings as needed.
If appropriate, save changes to the styles in the style library. Otherwise, the changes are only saved in the template file.
 - Customize the default sheet and add any other sheets that you want in new drawings.
 - Set the properties for the drawing file.
- 3 Copy drawing resources:
 - To copy a drawing resource from one template or drawing to another, right-click the resource in the browser, and then select Copy. Paste the resource to the appropriate folder in the browser of the other drawing.
 - Use the Drawing Resource Transfer Wizard to copy drawing resources from a source drawing to one or more target drawings. Close Autodesk Inventor, and then click Start menu ➤ Programs ➤ Autodesk ➤ Autodesk Inventor [version] ➤ Tools ➤ Drawing Resource Transfer wizard.
- 4 Place the default base views and projected views. If the file contains one or more base views, you are prompted to locate a component file.
- 5 Save the file in the Autodesk\Inventor[version]\Templates folder or a subfolder of Templates. A drawing file automatically becomes a template when it is saved to the Templates folder.

The location of the Templates folder is by default:

Microsoft Windows XP: Program Files\Autodesk\Inventor [version]\Templates.

Microsoft Vista: Users\Public\Documents\Autodesk\Inventor [version]\Templates.

Create a DWG template in Autodesk Inventor from an IDW template

- 1 In Autodesk Inventor, click  ➤ New
- 2 Select an .idw template and click OK.
- 3 Click  ➤ Save As ➤ Save Copy As.
- 4 Select the Save as type drop-down menu, and select Inventor Drawing Files (.dwg).
- 5 Save the file into your templates folder, as defined in Application Options or in your project. This template is available each time you create a drawing.

Create a .dwg template in Autodesk Inventor from an AutoCAD .dwg template

If you must deliver a DWG file from Autodesk Inventor that is fully editable in AutoCAD, consider using translation to export an AutoCAD drawing. You can leverage DWG TrueConnect to create drawings in Autodesk Inventor that translate more accurately and require less cleanup in AutoCAD to meet strict formatting standards.

Step 1: Open the Template

- 1 In Autodesk Inventor, click  ➤ Open. Select the Files of type drop-down menu, and select AutoCAD drawings (*.dwg). Select the AutoCAD DWG file that you want to use as a template.

NOTE If your AutoCAD file is an AutoCAD drawing template (.dwt), you must rename a copy as a DWG file (.dwg) before opening it in Autodesk Inventor.

- 2 Click Options. In the File Open Options dialog box, verify that the Open option is selected, and click OK.
- 3 With the file open, use Save As (not Save Copy As) to save it as an Autodesk Inventor drawing in your templates folder.

The location of the Templates folder is by default:

Microsoft Windows XP: Program Files\Autodesk\Inventor [version]\Templates.

Microsoft Vista: Users\Public\Documents\Autodesk\Inventor [version]\Templates.

NOTE You can also specify a templates folder in your project file.

Step 2: Create a New Standard

- 1 In Autodesk Inventor, click Manage tab ► Styles and Standards panel ► Styles Editor to open the dialog box.
- 2 In the style list browser, expand Standard at the top of the list and right-click the Default Standard (ANSI) style. Select New Style from the context menu.

NOTE The new Standard style is based on the style you select.

- 3 Name the new standard style and click OK.
- 4 In the style list browser, double-click the new standard to activate it.
- 5 In the style list browser, expand Object Defaults and right-click the Object Defaults (ANSI) style. Select New Style from the context menu.
- 6 In the New Style Name dialog box:
 - 1 Enter a unique name for the new style.
 - 2 Verify that Add to Standard is checked and click OK.
- 7 In the style list browser, right-click the old standard style and select Purge Style and Sub Styles. Click Yes in the Purge Unused Styles and Sub-Styles dialog box.

NOTE This action removes the old Standard and object defaults styles and helps avoid confusion when editing the styles in the next step.

Step 3: Edit the Standard

- 1 Select the new standard in the style list browser.
- 2 Click the Available Styles tab.
- 3 Select Dimension in the Style Type list:
 - 1 Check the box next to the dimension styles in the template you want to include in the standard.
 - 2 Clear the check box next to any dimension styles you do not want to use in this template.
- 4 Select Layers in the Style Type list:
 - 1 Check the box next to the layers in the template you want to include in the standard.
 - 2 Clear the check box next to any layers you do not want to use in this template.
- 5 Select Text in the Style Type list.
 - 1 Check the box next to the text styles you want to include in the standard.
 - 2 Clear the check box next to any text styles you do not want to use in this standard.
- 6 Click Save to save the edits.

Step 4: Edit the Object Defaults Style

- 1 In the style list browser, Select the new Object Defaults style.
- 2 Find the Linear Dimension object in the Object Type list.
- 3 Click the corresponding value in the Object Style column to change the default dimension style for linear dimensions to one of the dimension styles from the template.
- 4 Click the corresponding value in the Layer column to change the default layer to one of the template layers.
- 5 Repeat steps- 4 to remap dimension and text objects to use template styles and layers. Remap the rest of the objects (those objects that are not dimensions or text) to use template layers. These objects do not have

corresponding object styles in AutoCAD, so their object styles can be left alone.

- 6 Click Save to save the Object Defaults style. Click Done to close the Styles Editor.

Step 5: Purge Unused Styles

In Autodesk Inventor, click Manage tab ➤ Styles and Standards panel ➤ Purge to remove any extra, unused styles from the template.

In the Purge Styles dialog box, for each style, click the corresponding value in the Purge? column to select or deselect the style to purge.

NOTE If there are styles you want to keep but you did not select them in Object Defaults, select them on the Available Styles tab in the standard to prevent them from being purged.

(Optional) Save to the Styles Library

In Autodesk Inventor, click Manage tab ➤ Styles and Standards panel ➤ Save to save the styles in the template to the Styles library for future use and sharing.

NOTE Since you created a new standard style and object defaults style, you do not receive styles conflict messages when using this template if you do not save to the library.

Step 6: Save and Close the Template

Use Save As or Save Copy As to save the file in your templates folder, as defined in Application Options, or in your project. This template is available each time you create a drawing.

NOTE Naming the file standard.dwg makes it the default template. Rename the previous default template to avoid deleting it.

Translation Tip:

During translation, if you specify the original AutoCAD template or sample file, any changes made to layers or dimension and text styles in the source drawing in Autodesk Inventor is reset to the original template settings in the translated file.

NOTE

- When you create a drawing from a template, all AutoCAD data is removed, except for block definitions. All block instances remain on any sheet (excluding the model space) in a template. You can place AutoCAD title blocks or borders on a sheet and use them in Autodesk Inventor. Any AutoCAD data that needs to remain on a sheet must be placed into a block.
 - When using a style library, style definitions are refreshed from the library when creating a file using a template.
-

Customize drawing templates

Customize drawing templates



The following steps get you started quickly with your company standard for drawing files. These steps are overview of the main components of a customized drawing template.

The Drawing Resources folder in the Drawing browser contains folders for sheet formats, title blocks, borders, and sketched symbols. Each can be reused in new sheets and saved in the template file.

Custom IDW template setup overview

- 1 In Autodesk Inventor, open a new drawing (.idw) file.
- 2 Set template options in the Tools tab ➤ Options panel ➤ Document Settings dialog box.
- 3 Preparation: Make sure Sheet:1 is active, and then delete the Border and Title Block.
- 4 Create a Border.
- 5 Design a Title block. Add properties and format the text.
- 6 Create a Sheet Form as needed.
- 7 Save the file in the default Template folder.
- 8 Add note text, revision tables, and sketch symbols to the sheet as needed.

To make a template the default, save it in the Templates folder with the file name standard.idw:

Microsoft Windows XP: Program Files\Autodesk\Inventor [version]\Templates.

Microsoft Vista: Users\Public\Documents\Autodesk\Inventor [version]\Templates.

NOTE Alternatively, you can specify a different template location for each project in the Projects dialog box.

To avoid replacing the existing default template, move or rename the existing standard template before saving the new template. Files that reside in the Templates folder appear on the Default tab of the New dialog box when you create new files.

NOTE To add tabs to the new dialog box, create new subfolders in the Templates folder, and add template files to them. The New dialog box displays a tab for each subfolder in the Templates folder.

Configure Document Settings

Use the Tools tab ➤ Options panel ➤ Document Settings dialog box to set the options in the templates.

TIP You can copy iProperties from one Autodesk Inventor file to your template by selecting Copy Model Properties, and then selecting the desired iProperties in the Properties dialog box. Copied iProperties are not associative so do they do not update when the source file is updated.

Clean up the browser


For optimum workflow, create the title block formats and any custom borders and sketched symbols before creating the templates, so that they are available when you set up the sheet formats.

- 1 Right-click the Default Border under Sheet:1 in the browser, and select Delete.
- 2 Perform the same action on the Title Block, so that the Sheets folder is empty.

Setting the border

The default border is parametric. It automatically adjusts its size and labeling when the sheet is resized. All sheets using that border automatically update to display your changes when you edit a default border definition.

Custom borders can be created and saved in the current drawing. To create a custom border, right-click Borders, and select Define New Border. To make a user-defined border parametric, use the General Dimension command to define the distance from each side of the border to the theoretical sheet edge.

- 1 To customize a default border, in the browser, right-click Sheet :1, and select Edit Sheet. The sheet displayed is the default sheet from the Standard.idw file.
- 2 Click the drop-down list, select the sheet size that to be your default.
- 3 Select a default orientation, and click OK to save and close.
- 4 Expand the Drawing Resources folder and then expand the Borders folder.
- 5 Right-click the Default Border, and select Insert Drawing Border. The Default Drawing Border Parameters dialog box displays.
- 6 Click More  on the bottom right corner of the Default Drawing Border Parameters dialog box to expand it for more settings.
- 7 Set the border parameters to your company specifications. Click OK to save and close the dialog box.

NOTE The input value for Number of Zones must be even numbers. Alternately, you can use your custom border as described previously.

You cannot edit the default border after it is placed. To change the border, delete it and insert a new border with the appropriate properties.

Title block basics - Part A: Set up and maintenance tips

The title blocks used in an Autodesk Inventor drawing are dynamic and include information about the drawing, the sheet, and the design properties. The title block automatically updates to display current information, as information changes.

Dynamic information in a title block is referred to as a property field and is added through the text command. The information can be entered through the Drawing Properties dialog box or through user input (as prompted entries).

- 1 Right-click the ANSI-Large title block under Sheet:1 in the browser, and select Delete from the context menu.
- 2 Expand the Drawing Resources folder, and then the Title Blocks folder.
- 3 To base your title block on an existing one, right-click the title block and select Edit. To create a title block, select Manage tab ➤ Define panel ➤ Title Block.
- 4 To insert an AutoCAD title block, select the AutoCAD icon (the Define new title block must be active).

TIP For better visibility, turn off the Grid Lines in Tools tab ➤ Options panel ➤ Application Options. Select the Sketch tab, and cancel the Grid Lines selection.

- 5 Use the sketch commands to modify the title block, dimension, or to place property fields. (See Setting Property Fields in the following section.)
- 6 Insert a .bmp (only one allowed) into the title block by selecting the Insert image command, and then browsing for the image.
- 7 After completing the title block, right-click and select Save Title Block. In the Save Edits dialog box, click Save As and save a new custom title block to create a title block, or click Yes to save edits to a pre-existing one.
- 8 Double-click the title block to insert it into the drawing.

NOTE Dimensions used to set the size of elements in a title block or border are hidden upon exiting sketch mode.

You cannot directly modify the title block on a drawing sheet. To change a title block, edit the title block definition that you used to create it. Editing a definition changes all title blocks in the file you are editing from it.

NOTE To copy a titleblock from one template or drawing to another, right-click the titleblock in the browser and select Copy from the menu. Paste to the titleblock folder in the browser of the other drawing.

Use the Drawing Resource Transfer Wizard to make a change in one template and apply the changes to any existing drawing files. It comes in handy in many circumstances. For example, you can use the Drawing Resource Transfer Wizard to distribute a company area code on a title block that was used to

create many drawings by updating the existing title blocks in selected drawings with the new area code.

Title block basics - Part B: Adding Properties

Use the Text or Edit Text commands to set or change the values of Properties (including prompted entries) in a title block, border, or sketched symbol.

NOTE In drawings, the default text format is controlled by the active object default style of the standard. To change the default text format for a drawing or template, modify the text style and object defaults style using the Styles Editor.

- 1 On the ribbon, click Sketch tab ➤ Draw panel ➤ Text.
- 2 Left-click or window the position to insert the property field.
- 3 In the Format Text dialog box, select the Type of category of properties from which to choose.
- 4 Prompted Entry specifies a prompt to enable entry of information at the time that a title block is placed. Select Prompted Entry to prompt the user for information when inserting the template or the title block.
 - Select the field name in the Edit window, and then enter the appropriate prompt. This is like AutoCAD attributes.
- 5 Select a property from the drop-down to display in the title block.
- 6 Add text or symbols to the annotation by placing them in the edit box. The value you select must be placed between the arrows.
- 7 Select the contents displayed in the edit box and apply any of the format properties available.

NOTE Use Fit Text to sizes the text to fit the designated space, such as a text box. Use Stretch to specifies the text width. For example, enter **100** to display the text as designed, enter **50** to decrease the width of the text by 50%. Single line text removes all line breaks from multi-line text.

Title block basics - Part C: Format text tips

The Format Text dialog box provides several formatting features that can be applied to sketch text:

- Use the text box to apply constraints and dimensions to text.

- Use the Fit text feature to sizes the text to fit the designated space, such as a text box.
- Use the Stretch feature to specify the text width. For example, enter **100** to display the text as designed, enter **50** to decrease the width of the text by 50%.
- The Single line text feature removes all line breaks from multiline text.
- After completing your title block, right-click and select Save Title Block. In the Save Edits dialog box, select Save As to save a new custom title block or Yes, if you edited a pre-existing one.

Setting Sheet Formats

A sheet layout contains a sheet format, border, and a title block. You can edit the information in the default sheet layout for the sheet name, sheet size, orientation, count attribute, and print attribute. The default sheet format is the last layout used or edited in the drawing template.

You can define one or several sheet formats and then add them to the Drawing Resources folder. Once you add a sheet format to a drawing, you can use it to add new sheets to that drawing.

- 1 Add a new sheet to the drawing, using either the default sheet or one of the existing sheet formats.
- 2 Set the size and orientation for the sheet.
- 3 Add the standard components to the sheet, including a border, title block, and standard views.
To be included in the format, views must be completely within the border of the sheet.
- 4 Right-click the sheet and select Create Sheet Format from the menu.
- 5 Enter the name for the new sheet format in the edit box. Click OK to save and exit the dialog box.

Sheet formats are added to the Drawing Resources folder in the browser. To add a sheet using the new format, expand Drawing Resources ► Sheet Format, and then double-click the appropriate sheet.

NOTE A sheet format cannot be edited directly. To change an existing sheet format, use it to create a sheet, delete the unwanted format, then use the sheet to create a format with the same name.

Sheet formats contain a predefined border, title block, and views. The format corresponds to a standard sheet size with an appropriate title block and border.

Updating styles

You can create new styles or rename styles in a document, but they affect only that document. Take a separate step to save a new or changed style to a style library because doing so replaces the library version. The CAD Administrator can control access to the style library.

Configure for optimal performance

Performance refers to how long it takes Autodesk Inventor to complete an operation on your computer, including loading files. Capacity refers to how much memory is required to do an operation combined with how much memory is used by your computer at any single moment.

Ensure your designers are aware of how to increase performance and manage capacity when working with large files. Enforce these practices to increase productivity.

For detailed information, refer to the Inventor online help in the following section:

- 1 In Autodesk Inventor, click Help.
- 2 Select the Contents tab, and navigate to the Help topics:
- 3 Autodesk Inventor ► CAD Manager Fundamentals ► Configure for optimal performance ► [Work smart in assemblies](#) on page 141

IMPORTANT Remember to click **Back** to return to the CAD Manager Guide if you click any of the links on this page.

Also refer to: [Increase performance and capacity](#) on page 32

Work smart in assemblies

Here are some suggested practices that will speed up and simplify assembly modeling.

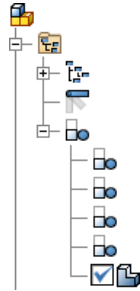
Customize assembly viewing

- As you add components, turn off the visibility of components that no longer impact the portion of the design on which you are currently focused.
- Pause the cursor over a component in the browser to highlight it on the graphics window.
- Right-click, and then select Find in Browser or Find in Window to locate components.
- Click the binoculars symbol on the browser menu bar to activate the Find dialog box. Specify search criteria to locate constraints, features, and files by criteria specific to the search object.
- Turn off enabled status for parts you do not need to select but need to see for frame of reference.
- Use color to segregate subsystems in an assembly. For example, display all components in the pneumatics system in one color, all components supplied by a certain vendor in another color, and so on. Save the colored subsystems as a Design View and copy to a Level of Detail to manage the subsystems visibility.

Use representations to customize component statuses

- Save a Design View or Level of Detail representation of a complex subassembly and turn on visibility for only the components needed to place the subassembly. Use a Level of Detail representation when you place the subassembly.
- Turn off visibility of nonessential components and save the Design View representation with a unique name. Reload the Design View representation whenever you work on the assembly.
- Copy a Design View representation to a Level of Detail to manage the visibility and suppression status.
- Create and save Positional representations to capture "snapshots" for motion studies and evaluation of an assembly in various positions.
- Create overlays in drawings of Positional representations to show the assembly in various extensions.
- Create Level of Detail representations to avoid loading unneeded components into memory. Improve computer performance with lower memory requirements.

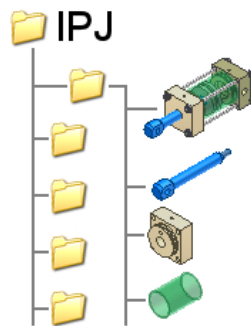
- Create a Substitute Level of Detail using a surface composite Shrinkwrap or Derived part to significantly reduce file size and memory consumption.



- When opening an assembly, specify a mix of representations to achieve the appropriate working view and modeling efficiency.

Plan for efficiency

- Plan the top-level assembly and subassembly structure before you create parts. Hierarchy improves performance.
- Create logical subassemblies and combine them into larger assemblies. Use Promote and Demote as required to assist in creating subassemblies.
- Keep all components used in a subassembly in the same directory.



- Create a shared network directory for components that will be shared by many designers on many projects.
- Assign the Summary and Project properties for individual components.

- Create a unique template and use it to create components for a specific project or subassembly. Predefine common properties in the template so all components created from that template inherit the properties.
- Search for attributes both inside and outside Autodesk Inventor to find needed component files.
- Save and name attribute searches that you are likely to use again.

Manage assembly constraints

- Use only as many constraints as are needed to control component position and movement. Constraints consume memory.
- Consider using Grip Snap to position and then ground components that do not need to move.
- Avoid redundancy. In the Application Options dialog box on the Assembly tab use Enable constraint redundancy analysis to check for redundant constraints. Delete redundant constraints. Turn off the option after completing the analysis.
- Ground at least one component in each assembly to optimize solve performance.
- Use a common constraint reference if possible. Constraining all components to a common component or geometry improves performance and reduces complexity. For example, use the Origin work features to constrain components in an assembly whenever possible.
- Constrain symmetrical assemblies to mid-planes and center axes.
- Use iMates to reduce overhead and enforce consistency.
- To locate a constraint, select the Model browser filter to collect all constraints in a browser folder.
- Pause the cursor over a constraint in the browser to highlight it in the graphics window.
- Avoid constraints between features that might be removed later in the design process.
- Pause the cursor over a constraint icon in the browser to show all property information for the highlighted constraint.
- Start constraining components by mating planar faces, and then add tangent, angular, and flush constraints.

- Pause the cursor over an exclamation point icon in the browser to show information about an unsolvable constraint.
- Repair model errors before you delete or change constraints.
- Locate and fix or suppress any constraint errors. Use the Design Doctor to isolate components.

Find constraint errors

Constraints with errors are marked with yellow symbols.

- Use the Modeling option of the browser view menu to gather all constraints into a folder at the top of the browser hierarchy. Expand the folder to see all constraints in the assembly.
- Highlight a constraint error to see the constrained parts highlight, and then zoom to the constraint in the graphics window.
- Use the Position option of the browser view menu to locate and highlight a single constraint error. Right-click to use the context menu to automatically expand the browser and reset browser focus to the selected constraint's other half.
- Use the Design Doctor to diagnose and suggest treatments for constraint errors. Use an Isolate treatment option to show only the affected components.

Organize design files in a folder structure

Avoid storing all assembly files in one folder. When Autodesk Inventor must search through hundreds or thousands of files in a single folder to find the correct file, it can take longer to open files. As part of the file name, Autodesk Inventor stores the path to the folder where a file is located. When there are fewer files in a folder, the file is located and opened more quickly. If possible, place all subassembly components in the same directory.

Set up Content Center

Autodesk Inventor Content Center libraries provide standard parts (fasteners, steel shapes, shaft parts, and so on) and features.

The Content Center configuration is a part of Project settings, and controls the internal setting of libraries within Content Center. You change the Content

Center configuration in the Configure Libraries dialog box accessed from the Project dialog box.

Refer to online help, for example:



- 1 In Autodesk Inventor, click Help.
- 2 Select the Contents tab, and navigate to the Help topics:
- 3 Autodesk Inventor ► Assemblies ► Content Center.
- 4 Click Content Center Installation and Setup. The Concept tab provides an excellent overview for setting up Content Center libraries: [Content Center installation and setup](#) on page 152

NOTE Content Center protects content with permissions. Use the Autodesk Server Console to create editing permissions for users who must modify content center libraries. Users requiring only read-only access do not need to be defined.

- 5 [Click here](#) on page 146 to learn how to share libraries in a networked environment.

In addition, see the published guide:
<http://www.autodesk.com/inventor-whitepapers>

IMPORTANT Remember to click **Back** to return to the CAD Manager Guide if you click any of the links on this page.

Share libraries across design workgroups

Before beginning, designate an administrator who installs and manages Content Center libraries on the Autodesk data management server.

Library configurations for team members are saved in the Autodesk Inventor project and can be edited in Project Editor. Each team member can create a personal project using the project file (.ipj) as a template, or include a common project (.ipj) file that is configured with the shared libraries. This enables each team member to have the same configuration.

NOTE You cannot include a project file in an Autodesk Vault project file. Autodesk Vault does not support included project files.

If you are connecting to a vault database and the Content Center libraries database, the administrator creates the vault project file first and makes it the active project. Then the administrator configures the Content Center libraries in the Configure Libraries dialog box.

Overview

- Install the server and client components for a design team.
- Create a common project file (.ipj) with the desired configuration of Content Center libraries. This project can be used as a template or as an included project.
- In the common project, change the folder where standard components are created, if needed.
- In Autodesk Data Management Server Console, create the user accounts for anyone requiring editing permissions. Communicate user account information and the Login procedure to each member in the team.

Install the server components

Install Content Center libraries on the server. For installation details, please refer to the Autodesk Vault Server Implementation Guide. The guide is installed as a PDF in Program Files ➤ Autodesk ➤ ADMS [version] ➤ Help.

TIP If appropriate, use two separate servers for Content Center libraries and Autodesk Vault files. Install the server on two computers, and then use one as a Vault server, and the other as a Content server.

Use the server console to configure libraries on the server

Before you start using Content Center, set up Content Center libraries to correspond with your needs. Use the server console to create, attach, import, export, detach, and delete Content Center libraries, and to change a library status.

- 1 To run the server console, click Start ➤ Programs ➤ Autodesk ➤ Autodesk Data Management ➤ Autodesk Data Management Server Console [version].
- 2 In the server console window, expand the root folder and perform the appropriate changes:
 - To create a library, right-click the Libraries entry and choose Create from the menu.

- To attach a library saved in the default libraries folder, right-click the Libraries entry and choose Attach Library from the menu.
- To import a library, right-click the Libraries entry and choose Import Library from the menu.
- To detach a library from the server, right-click the library name and choose Detach from the menu.

Create Content Center Editor user accounts in the server console

Create Content Center Editor user accounts for anyone requiring editing permissions.


- 1 Run the server console: Program Files ► Autodesk ► Autodesk Data Management ► Autodesk Data Management Server Console [version].
- 2 Click Tools ► Administration.
- 3 In the Administration dialog box, click Users.
- 4 In the User Management dialog box, click New User.
- 5 In the New User dialog box, enter the appropriate information for the user.
- 6 Click Roles to assign the user a Content Center role.
- 7 Select Content Center Editor to assign editing permissions.
- 8 Click Vaults to assign the user to a vault.
- 9 If needed, click Groups to assign the user to a group (optional).
- 10 Click OK.
- 11 In the User Management dialog box, and then in the Administration dialog box, click Close.
- 12 Communicate user account information and the Login procedure to each member in the team.

Install the client components

On the local computers, install Autodesk Inventor. Select the Use Vault Server option to skip installing Content Center libraries.

Connect to the server from Autodesk Inventor


- 1 Start Autodesk Inventor.


- 2 Click  ➤ Vault Server ➤ Log In.

- 3 Enter the appropriate data to the Log In dialog box:

- User Name: *Enter the user name.*
- *Enter the password.*
- Server: *Enter the name of the computer (for example: MyRemoteServer), the IP address (for example: 141.111.111.111), or the full computer name for your network domain (for example: pc-name.xyz.autodesk.com).*
- Database: *Enter the name of a Vault database located on the specified server.*
- Select Automatically login next session to get automatically logged in the next Autodesk Inventor session.

TIP If you use two servers for Content and Vault, change the setting on

computers of the workgroup members: Click  ➤ Vault Server ➤ Connection Options and select Use Separate Servers for Content and Vault in the Connection Options dialog box. To log in to the Content Center server,

choose  ➤ Vault Server ➤ Content Center Log In. To view the

status of the server, choose  ➤ Vault Server ➤ Connection Status.

Create a project file (.ipj) and configure Content Center libraries


Library configurations are saved in the Autodesk Inventor project and edited in Project Editor.

After the common project is configured, copy the project file (.ipj) to a shared location. Request each team member to use the project as a template or as an included project.

TIP You can create several projects with different setting that is specific for your company projects.

Configure Content Center libraries



- 1 Click  ► Manage ► Projects.
- 2 In the Projects dialog box, create a project file or select an existing project file.
- 3 In the Projects dialog box, click the Configure Content Center Libraries command.
- 4 In the Configure Libraries dialog box, configure Content Center libraries. Remove all libraries you do not use and add libraries you want to use. If needed, migrate old libraries.
- 5 Click OK to close the Configure Libraries dialog box.
- 6 Click Save in the Projects dialog box.

NOTE To increase performance, remove all Content Center libraries that are not used by the workgroup. In the Configure libraries dialog box select a library and click Remove Library. If needed, removed libraries can be added back using the Add Library command.

Optionally, change the folder where standard components are created.

When Content Center creates a component, it first searches a cache of previously utilized content located in the Content Center Files location to improve access time to commonly used components. Although it is possible to set different Content Center Files location, we recommend that you use the same Content Center File location for all projects to optimize the performance.

- 1 In the Project dialog box, expand Folder Options.
- 2 Right-click the Content Center Files entry, and enter the new location of the Content Center Files folder.
 - Specify a network location for all users to share the same cache memory of instanced components. The network location ensures that everyone in the shared environment is always accessing the same standard parts and automatically gets the most recent library content.
 - Specify a location on local drives for users to decide when they want to refresh their libraries. In this scenario, a user must click Manage tab ► Content Center panel ► Refresh to get the latest content.

- 3 Click Save and Done in the Projects dialog box.

Configure local project settings

Connect to the server to change the library configuration in the Autodesk Inventor project.

Use one of the following methods to create a local project:

Create a local project which uses the common project as an included file

NOTE You cannot include a project file in an Autodesk Vault project file. Autodesk Vault does not support included project files.

- 1 In Autodesk Inventor, in the Project dialog box, create a new personal project file.
- 2 Double-click the new project to make it the active project.
- 3 In the Project dialog box, highlight Included files, and then click the Edit selected item.
- 4 Browse to the location of the common project file (.ipj) configured by the administrator.
- 5 Click Open.
- 6 Click Save and Done in the Projects dialog box.

Use the common project as a template for a local project

If you create a personal project file with the shared project as a template, you can save filters and favorites locally.

TIP Select the common project before you click New to use the selected project as a template. The new project has the same library definitions as the common project.

- 1 In Autodesk Inventor, in the Project dialog box, add the common project file (.ipj) created by the Administrator to your projects:
 - 1 Click Browse.
 - 2 In the Choose Project file dialog box, select and open the common project file (.ipj) created by the administrator.
- 2 In the Project dialog box, highlight the common project file.

- 3 Click New.
- 4 Create a new personal project file.
- 5 Click Finish to create the project.
- 6 If appropriate, edit the new project.
- 7 Double-click the personal project to make it the active project.
- 8 Click Save and Done in the Projects dialog box.

Content Center installation and setup

You can work with Content Center as a stand-alone user (libraries are saved in the Desktop Content location on your computer) or as a member of a workgroup (libraries are shared on a Vault server).

If you select the Desktop Content option in the installation, Content Center libraries are installed to the folder specified in the installation configuration. After the installation, the Desktop Content environment is set up and Content Center is ready for use. If you select the Vault Server option in the installation, Content Center libraries are not installed on your computer. You must install Content Center libraries on the server, and log in to the server to use Content Center.

TIP Migrate your legacy user libraries to use them with the new version of Autodesk Inventor.

Before you start using Content Center, set up Autodesk Inventor and Content Center libraries to correspond with your needs:

- Set the Application Options. The Content Center tab in the Application Options dialog box sets the storage location of Content Center libraries and refreshing of standard parts on instancing. The File tab in the Application Options dialog box specifies the default Content Center Files folder where instanced Content Center parts are saved.
- Set Content Center options in the Autodesk Inventor project. Use the Configure Libraries dialog box to configure Content Center libraries. If appropriate, change the Content Center Files folder.

After a successful installation and configuration of Autodesk Inventor Content Center, the following components are available:

- Standard Content Center libraries installed in the Desktop Content folder or on an Autodesk Vault Server and configured in the project.
- One or more user libraries located in the Desktop Content folder or on an Autodesk Vault Server and configured in the project.

Content Center environment for a stand-alone user

If you want to work as a stand-alone user, perform the following steps to set up Content Center:

- Select the Desktop Content option in the Autodesk Inventor installation.
- If your legacy user libraries reside on the server, transfer them to Desktop Content location. Autodesk Inventor can do the transfer automatically on the first startup. Click Yes in the Content Center Library Migration dialog box.
- If appropriate, copy user libraries from an external Desktop Content location to your Desktop Content folder.
- Set up the configuration of Content Center libraries in the project:
 - Transfer user libraries from a server by using Library Transfer guide.
 - Select the In Use box to add libraries in the configuration.
 - Migrate and synchronize legacy user libraries by using the Update Tool.

TIP Legacy and new version of Autodesk Inventor on the same computer:

Use separate folders for old and new Desktop Content libraries. Remember that migration in Update Tool irreversibly changes user libraries. You cannot use migrated Desktop Content libraries in an old version of Autodesk Inventor.

Content Center environment for a design workgroup

You can eliminate the need for each member of a design workgroup to install and synchronize their own Content Center libraries by installing Content Center libraries on a Vault server.

Local settings for workgroup members

After Content Center libraries are set up on the server, members of the workgroups configure their local settings.

We recommend that you designate an administrator to install the server, set up projects and predefine the standard set of libraries to use in a design project. The administrator can create a common project and add shared libraries to the Content Center configuration, and then share the common project file (.ipj) with the other team members.

To log into the server within Autodesk Inventor, you specify the location (or name) of the remote server in the Log In dialog box.

Storage folder for instanced standard components

When you insert a Content Center library part in an assembly, the library part is copied from the server to the Content Center Files folder. The configuration of the Default Content Center Files folder is saved in Application Options, and the setting can be overridden in the project.


Set the Content Center Files folder depending on how you want to consume changes to the Content Center libraries:

- Specify a network location of the Default Content Center Files folder on each of the local computers for each user to get the most recent library content automatically.
- Specify a location on the local computers for users to decide when to update their libraries. In this scenario, use the Refresh Standard Components command to get the latest content.


Separate servers for Content Center and Vault


The Vault server manages libraries for Content Center and file databases for Autodesk Vault. If needed, you can use two separate servers for Content Center libraries and Autodesk Vault files. Install the server on two computers, and then use one as a Vault server and the other as a Content server:

- To enable the connection to two separate servers, change the Autodesk

Inventor settings on computers of the workgroup members: Click 

➤ Vault Server ➤ Connection Options and select Use Separate Servers for Content and Vault in the Connection Options dialog box.

- To log in to the Content Center server, click  ➤ Vault Server ➤ Content Center Log In.

- To view the status of the server, click  ➤ Vault Server ➤ Connection Status.

Comparing Desktop Content and Vault Server

	Desktop Content	Autodesk Vault Server
Installation	Autodesk Inventor installation includes Desktop Content libraries	Autodesk Inventor without libraries installed on client computers, Vault Server and libraries installed on the server
Additional software needed	No	IIS or AWS, SQL Server
User accounts and permissions	No (can be added on the file access level of the operation system, if appropriate)	Yes (managed by Vault Server)
Supports multiple Autodesk Inventor versions on one computer	Yes (with no limitations)	Yes, with limitations (one version of Vault Server, the corresponding and two previous versions of Autodesk Inventor, Autodesk Inventor Client Update installed for legacy Autodesk Inventor versions)
Management of Content Center libraries	In Autodesk Inventor, by using the Configure Libraries dialog	External, by using Vault Server Console
Work with Content Center	No log-in or permissions needed, all Content Center functionality available	Connection and log-in to the server required, editing permissions needed to perform editing tasks
Migration of legacy user libraries	Yes (transfer from ADMS or copying from a legacy Desktop Content folder and migration by Update Tool)	Yes (migration and set up on the Vault Server, then migration by Update Tool)

Working with Autodesk Inventor 2009, 2010, and 2011 libraries on the same server

Autodesk Inventor 2011, the Autodesk server 2011, and the server Client Update installation for Autodesk Inventor 2009 and 2010 enable you to access

Autodesk Inventor 2009, Autodesk Inventor 2010 and Autodesk Inventor 2011 libraries on one library server.

To support Side-by-Side installation, the Autodesk server maintains Content Center libraries that were built for different Autodesk Inventor versions. Libraries are saved in the corresponding Partition of the Autodesk server. Libraries in Autodesk Inventor 2008 partition are used by Autodesk Inventor 2009 with Client Update patch. Libraries in Autodesk Inventor 2010 partition are used by Autodesk Inventor 2011 with Client Update patch. Libraries in Autodesk Inventor 2011 partition are used by Autodesk Inventor 2011.

The default prefix of the library name is determined by the partition where the library is saved. You can copy user libraries from Autodesk Inventor 2009 or 2010 partition into Autodesk Inventor 2011 partition. Copying from Autodesk Inventor 2011 partition to Autodesk Inventor 2009 or 2010 partition is not supported.

Share libraries across design workgroups

Before beginning, designate an administrator who installs and manages Content Center libraries on the Autodesk data management server.

Library configurations for team members are saved in the Autodesk Inventor project and can be edited in Project Editor. Each team member can create a personal project using the project file (.ipj) as a template, or include a common project (.ipj) file that is configured with the shared libraries. This enables each team member to have the same configuration.

NOTE You cannot include a project file in an Autodesk Vault project file. Autodesk Vault does not support included project files.

If you are connecting to a vault database and the Content Center libraries database, the administrator creates the vault project file first and makes it the active project. Then the administrator configures the Content Center libraries in the Configure Libraries dialog box.

Overview

- Install the server and client components for a design team.
- Create a common project file (.ipj) with the desired configuration of Content Center libraries. This project can be used as a template or as an included project.
- In the common project, change the folder where standard components are created, if needed.

- In Autodesk Data Management Server Console, create the user accounts for anyone requiring editing permissions. Communicate user account information and the Login procedure to each member in the team.

Install the server components

Install Content Center libraries on the server. For installation details, please refer to the Autodesk Vault Server Implementation Guide. The guide is installed as a PDF in Program Files ➤ Autodesk ➤ ADMS [version] ➤ Help.

TIP If appropriate, use two separate servers for Content Center libraries and Autodesk Vault files. Install the server on two computers, and then use one as a Vault server, and the other as a Content server.

Use the server console to configure libraries on the server

Before you start using Content Center, set up Content Center libraries to correspond with your needs. Use the server console to create, attach, import, export, detach, and delete Content Center libraries, and to change a library status.

- 1 To run the server console, click Start ➤ Programs ➤ Autodesk ➤ Autodesk Data Management ➤ Autodesk Data Management Server Console [version].
- 2 In the server console window, expand the root folder and perform the appropriate changes:
 - To create a library, right-click the Libraries entry and choose Create from the menu.
 - To attach a library saved in the default libraries folder, right-click the Libraries entry and choose Attach Library from the menu.
 - To import a library, right-click the Libraries entry and choose Import Library from the menu.
 - To detach a library from the server, right-click the library name and choose Detach from the menu.

Create Content Center Editor user accounts in the server console

Create Content Center Editor user accounts for anyone requiring editing permissions.

- 1 Run the server console: Program Files ► Autodesk ► Autodesk Data Management ► Autodesk Data Management Server Console [version].
- 2 Click Tools ► Administration.
- 3 In the Administration dialog box, click Users.
- 4 In the User Management dialog box, click New User.
- 5 In the New User dialog box, enter the appropriate information for the user.
- 6 Click Roles to assign the user a Content Center role.
- 7 Select Content Center Editor to assign editing permissions.
- 8 Click Vaults to assign the user to a vault.
- 9 If needed, click Groups to assign the user to a group (optional).
- 10 Click OK.
- 11 In the User Management dialog box, and then in the Administration dialog box, click Close.
- 12 Communicate user account information and the Login procedure to each member in the team.


Install the client components

On the local computers, install Autodesk Inventor. Select the Use Vault Server option to skip installing Content Center libraries.

Connect to the server from Autodesk Inventor


- 1 Start Autodesk Inventor.




- 2 Click  ► Vault Server ► Log In.
- 3 Enter the appropriate data to the Log In dialog box:
 - User Name: *Enter the user name.*
 - *Enter the password.*
 - Server: *Enter the name of the computer (for example: MyRemoteServer), the IP address (for example: 141.111.111.111), or the full computer name for your network domain (for example: pc-name.xyz.autodesk.com).*

- Database: *Enter the name of a Vault database located on the specified server.*
- Select Automatically login next session to get automatically logged in the next Autodesk Inventor session.

TIP If you use two servers for Content and Vault, change the setting on

computers of the workgroup members: Click  ➤ Vault Server ➤ Connection Options and select Use Separate Servers for Content and Vault in the Connection Options dialog box. To log in to the Content Center server,

choose  ➤ Vault Server ➤ Content Center Log In. To view the

status of the server, choose  ➤ Vault Server ➤ Connection Status.


Create a project file (.ipj) and configure Content Center libraries

Library configurations are saved in the Autodesk Inventor project and edited in Project Editor.

After the common project is configured, copy the project file (.ipj) to a shared location. Request each team member to use the project as a template or as an included project.

TIP You can create several projects with different setting that is specific for your company projects.

Configure Content Center libraries

- 1 Click  ➤ Manage ➤ Projects.
- 2 In the Projects dialog box, create a project file or select an existing project file.
- 3 In the Projects dialog box, click the Configure Content Center Libraries command.
- 4 In the Configure Libraries dialog box, configure Content Center libraries. Remove all libraries you do not use and add libraries you want to use. If needed, migrate old libraries.
- 5 Click OK to close the Configure Libraries dialog box.

- 6 Click Save in the Projects dialog box.

NOTE To increase performance, remove all Content Center libraries that are not used by the workgroup. In the Configure libraries dialog box select a library and click Remove Library. If needed, removed libraries can be added back using the Add Library command.

Optionally, change the folder where standard components are created.

When Content Center creates a component, it first searches a cache of previously utilized content located in the Content Center Files location to improve access time to commonly used components. Although it is possible to set different Content Center Files location, we recommend that you use the same Content Center File location for all projects to optimize the performance.

- 1 In the Project dialog box, expand Folder Options.
- 2 Right-click the Content Center Files entry, and enter the new location of the Content Center Files folder.
 - Specify a network location for all users to share the same cache memory of instanced components. The network location ensures that everyone in the shared environment is always accessing the same standard parts and automatically gets the most recent library content.
 - Specify a location on local drives for users to decide when they want to refresh their libraries. In this scenario, a user must click Manage tab ➤ Content Center panel ➤ Refresh to get the latest content.
- 3 Click Save and Done in the Projects dialog box.

Configure local project settings

Connect to the server to change the library configuration in the Autodesk Inventor project.

Use one of the following methods to create a local project:

Create a local project which uses the common project as an included file

NOTE You cannot include a project file in an Autodesk Vault project file. Autodesk Vault does not support included project files.

- 1 In Autodesk Inventor, in the Project dialog box, create a new personal project file.

- 2 Double-click the new project to make it the active project.
- 3 In the Project dialog box, highlight Included files, and then click the Edit selected item.
- 4 Browse to the location of the common project file (.ipj) configured by the administrator.
- 5 Click Open.
- 6 Click Save and Done in the Projects dialog box.

Use the common project as a template for a local project

If you create a personal project file with the shared project as a template, you can save filters and favorites locally.

TIP Select the common project before you click New to use the selected project as a template. The new project has the same library definitions as the common project.

- 1 In Autodesk Inventor, in the Project dialog box, add the common project file (.ipj) created by the Administrator to your projects:
 - 1 Click Browse.
 - 2 In the Choose Project file dialog box, select and open the common project file (.ipj) created by the administrator.
- 2 In the Project dialog box, highlight the common project file.
- 3 Click New.
- 4 Create a new personal project file.
- 5 Click Finish to create the project.
- 6 If appropriate, edit the new project.
- 7 Double-click the personal project to make it the active project.
- 8 Click Save and Done in the Projects dialog box.

Set up the Content Center environment for a stand-alone user

If you work as a standalone user, we recommend you to work in the Desktop Content environment. You do not need to log in to a server or set any user permissions. Libraries are treated as files in the Desktop Content folder on

your hard disc. Work with libraries, for example creating, deleting, moving, or copying libraries, is more convenient.

Step 1: Install Autodesk Inventor


Select the Desktop Content option in the Autodesk Inventor installation. Then select Standard Content Center libraries to install. Libraries are installed in the Desktop Content folder on your computer.

Step 2: If appropriate, transfer legacy user libraries on the first startup

When you first run Autodesk Inventor, Content Center looks for legacy user libraries saved on your computer. If a legacy ADM Server is installed on your computer and legacy user libraries are found, Content Center automatically starts transfer of legacy libraries into the Desktop Content location. Click Yes in the Content Center Library Migration dialog box to perform the transfer.

Step 3: Configure Content Center libraries in the project



- 1 Click  ► Manage ► Projects.
- 2 In the Projects dialog box, click the Configure Content Center Libraries command.
- 3 Use the options in the Configure Libraries dialog box to customize the configuration, and then click OK.
- 4 Click Save and Done in the Projects dialog box.

NOTE The fewer libraries you configure in a project, the better Content Center performs.


Tips:

- Use Library Transfer guide to manually transfer libraries from a Vault server to the Desktop Content location.
- Copy external Desktop Content user libraries to your Desktop Content folder: Locate a library to copy, and copy it as a file to your Desktop Content folder. The library appears in the Libraries list of the Configure Libraries dialog box, and can be added to the configuration.

NOTE The copied library must be unique in the Desktop Content folder. That means it must have a unique File Name and internal library name. The internal library name is created when the library is created, and cannot be changed. If more libraries with the same internal name exist in the Desktop Content folder, they are duplicates even if they have different File Names and Display Names. Content Center configuration does not allow adding libraries duplicate to currently configured libraries.

Step 4: If needed, change the folder where standard components are saved

To change the Content Center Files folder in Autodesk Inventor project:

- 1 Click  ► Manage ► Projects.
- 2 In the Projects dialog box, expand Folder Options.
- 3 Right-click the Content Center Files entry, and select Edit. Then set the new location of the Content Center Files folder.
- 4 Click Save and Done in the Projects dialog box.

If appropriate, change the default folder where standard components are created:

- 1 On the ribbon, click Tools tab ► Options panel ► Application Options.
- 2 On the File tab of the Application Options dialog box, change the default folder for Content Center files.
- 3 Click Apply and OK to close the Application Options dialog box.

NOTE When Content Center creates a component, it first searches a cache of previously utilized content located in the Content Center Files location to improve access time to commonly used components. Although it is possible to set different Content Center Files location, we recommend that you use the same Content Center File location for all projects to optimize the performance.

Administrative tasks for Content Center

In a workgroup environment, Content Center libraries are stored on an Autodesk Vault server. Libraries must be installed and set up on the server before you configure them in the project.

Use the server console to set up libraries on the server. You can create, attach, import, export, detach, and delete Content Center libraries, or change a library status. You must have administrator permissions to change the library configuration.

To run the server console, choose Start ► Programs ► Autodesk ► Autodesk Data Management ► Autodesk Data Management Server Console [version]

TIP Use separate servers for Content Center libraries and Inventor files in the shared environment. Install the Vault server on two computers, and then use one as a Vault server for Inventor files, and the other as a server with Content Center libraries.

Administrator tasks in the server console

You can perform the following tasks in the server console:

- Create a library.
- Change the status of a user-created library to read-only or read/write.
- Attach a library which is saved in the default folder for Content Center libraries.
- Import a library from other than the default library folder. The imported library is copied into the server data location, and can be then attached.

NOTE Use Import to migrate user libraries created in previous versions of Autodesk Inventor (not older than R10 libraries), or to add external user libraries to the server configuration. The source library is not changed, therefore you can select the backup copy of your library as the import source.

- Detach a library to remove a library from the library server. The detached library is not deleted and can be attached later.
- Delete a library from the server memory storage.
- Import and export libraries.

NOTE An .mdf and an .ldf file comprise one library. They must be maintained as a pair for both Import and Export.

- Review configuration of the Content Center library server.

Tips:

- To modify the Content Center standard parts or features to meet your requirements, use the server console to create a custom read/write library. Then copy part or feature families to the custom library, and edit the families. All modifications are saved in the custom library. Additionally, create standard parts that you want to add in a Content Center library. Then publish them to the custom library.
- Set up the Side-by-Side environment to install Autodesk Inventor 2008, 2009, and 2010 on the same computer, and use Content Center in all three versions of Autodesk Inventor.

Configuration of Content Center libraries on the server

The installation provides the following libraries: ANSI, DIN, ISO, JIS, GB, other international standards. The Parker library contains parts used by Parker for tube and pipe fittings. The PEM Fasteners library includes mostly sheet metal fasteners.

Content Center libraries can be installed only on a computer with the Autodesk data management server. You can customize the Content installation and select libraries to be installed. You can later add a Content Center library which was not installed: Run the installation CD, and select the library to install.

If you do not use a library which was installed, detach the library by using the server console. The library is not deleted but is no longer available in Content Center. You can attach the library to the server later, if needed.

The default Content Center libraries are read-only, cannot be edited, and the read-only status cannot be changed.

Standard parts in Vault or other data management environment**Refreshing Content Center parts in assemblies**

- If Content Center files are saved in Vault or other data management system, the Refresh Standard Components tool must check out all out-of-date Content Center part files to replace them.
- Refreshed Content Center files remain checked out when the refreshing process finishes. You must check the files in manually.
- If the refreshing process cannot change the read-only status to read/write automatically, standard parts are not refreshed and remain out-of date. This problem appears if read-only standard parts are not saved in Vault,

or if Inventor does not recognize that standard parts are saved in Vault (a single-user Inventor project is active, or standard parts were added to Vault by different Vaulted project).

Storage location for Content Center files

- If you work in Vault (or in another data management system) environment, each member of the workgroup must use a local folder to store Content Center files. The local Content Center Files folder should be controlled by the data management system.
- Do not change the location of the Content Center Files folder while a project is active.

Migration from previous versions

The Autodesk Inventor installation wizard automatically detects previous Content Center or Vault databases. If Autodesk Inventor 2008 or 2009 read/write libraries are detected on the computer, the installer displays a dialog box. Select the import option to migrate the libraries for use. Additionally, all previous Vault databases are recognized for update.

Notes:

- Autodesk Inventor 2008 and 2009 libraries are retained. The migration imports and creates a library for Autodesk Inventor 2011.
- If you add additional customization to legacy libraries Content Center, the server console can be used to re-import the libraries under a new name.
- Autodesk Inventor 9 (and older) customized libraries can be imported through the project file (*.ipj) using the Import Library command.
- If the migration of the libraries is postponed during installation, use the server console (included when the server is installed) after the installation to perform migrations.

Side-by-Side environment

Side-by-Side environment enables you to store and access Content Center libraries from three latest versions of Autodesk Inventor on one Vault server. Libraries are saved in the corresponding partition on the server.

To set up the Side-by-Side environment, install the new version of the server and Inventor. Then continue with the following steps:

- 1 Copy user libraries to appropriate partitions on the server. Remember that each version of Autodesk Inventor can access only libraries included in the corresponding partition.
- 2 Install Client Update patch for older versions of Autodesk Inventor.
- 3 In Autodesk Inventor, edit or create projects. Configure Content Center libraries and set up the Content Center Files folders in particular projects.

NOTE You must use separate Content Center Files folders for each version of Inventor. Sharing the Content Center Files folder by different versions of Inventor would cause unpredictable replacing of standard parts in assemblies.

Manage Content Center libraries by using the server console

Use the server console to create, attach, import, export, detach, and delete Content Center libraries, and to change a library status.

Administrative tasks are performed on the computer hosting the Content Center libraries by using the Server Console. You must have administrator permissions to change the library configuration on the server.

TIP To run the server console, choose Start ► Programs ► Autodesk ► Autodesk Data Management ► Autodesk Data Management Server Console [version].

Create a library

- 1 Run the server console.
- 2 In the server console window, expand the root, and right-click the Libraries entry.
- 3 Choose Create Library from the menu.
- 4 Enter the library name in the Create library dialog box, and click OK.
- 5 A new library is created.
By default, the new library is read/write. To change the status of a user-created library, right-click on the library, and select Read Only.

Attach a library

You can attach a library to the server, if the library is saved at the same folder as the existing libraries. If the library you want to add is saved in a different folder the library must be imported.

- 1 Run the server console.
- 2 In the server console window, expand the root, and right-click the Libraries entry.
- 3 Choose Attach Library from the menu.
- 4 The Available MDF Files dialog box displays all available Libraries that may be attached.
- 5 Select the Library you want to attach, and click OK.

TIP Use Attach to reattach a library you have already detached.

Import a library

NOTE For more information on the libraries see the *Autodesk Data Management Server (version) Implementation Guide*. The guide is installed as a PDF in Program Files ► Autodesk ► ADMS [version] ► Help.

Scenario example: A vendor has created content for your use and has copied the .mdf and .ldf files to a CD. Use the Import command to copy these files from the CD to the location of the server on the computer hosting the libraries.

- 1 Run the server console.
- 2 In the server console window, expand the root, and right-click the Libraries entry.
- 3 Choose Import Library from the menu.
- 4 Click Browse to find and select the .mdf file. The .ldf file is imported automatically.
- 5 If appropriate, enter a name for the Library in the Library Name text field, and click OK.

Detach a library

Use the Detach command to remove a library from the library server configuration. The detached library is not deleted and can be attached later.

- 1 Run the server console.
- 2 In the server console window, expand the root and the Libraries folder, and right-click the library to detach.

- 3 Choose Detach from the menu.
- 4 Click OK in the confirmation dialog box to detach the library from the server.

Delete a library

Use the Delete command to delete a library from the memory storage. If you want only to remove the library from the library server, use the Detach command.

- 1 Run the server console.
- 2 In the server console window, expand the root and the Libraries folder, and right-click the library to delete.
- 3 Choose Delete from the menu.
- 4 Click OK in the confirmation dialog box to delete the library from the server.

Change the status of a user-created library

You can change the status only for user-created Content Center libraries. The libraries installed by default with Autodesk Inventor Content Center are read-only, and the status cannot be changed.

- 1 Run the server console.
- 2 In the server console window, expand the root and the Libraries folder, and right-click the library to edit the status.
- 3 Select Read Only from the menu to set the library to be read-only. Unselect Read Only to set the library to be read/write.

NOTE The icon displayed in front of the library name indicates the library status.

Copy a library to a partition

Use the Copy command to copy a library from one partition to another.

NOTE To support Side-by-Side installation, the Autodesk server maintains Content Center libraries that were built for different Autodesk Inventor versions. Libraries are saved in the corresponding Partition of the Autodesk server. Libraries in Autodesk Inventor 2008 partition are used by Autodesk Inventor 2008 with Client Update patch. Libraries in Autodesk Inventor 2009 partition are used by Autodesk Inventor 2009 with Client Update patch. Libraries in Autodesk Inventor 2010 partition are used by Autodesk Inventor 2010.

- 1 Run the server console.
- 2 In the server console window, expand Libraries, and right-click the library to copy.
- 3 Choose Copy from the menu.
- 4 In the Copy Library dialog box, enter the display name, and choose OK. The library is copied into Autodesk Inventor 2011 partition.

NOTE Legacy user libraries are imported into Autodesk Inventor 2008 or 2009 partition. To use legacy libraries in Autodesk Inventor 2011, you must copy them into the Autodesk Inventor 2011 partition on the server, and then update them by using the Update Tool. For more information, see the Update Content Center libraries procedure.

Quick Reference

Autodesk Server Console - Reference

Use the server console to perform Content Center Library administrative tasks.

The server console displays the list of Content Center libraries currently available on the server.

NOTE An .mdf and an .idf file comprise one library and must be maintained as a pair for both Import and Export.

Access

Start ► Programs ► Autodesk ► Autodesk Data Management ► Autodesk Data Management Server Console [version].

Context Menu for Libraries folder

Create Library Creates a read/write Content Center library that can be configured for use in a local or remotely served Autodesk Inventor project.

Attach Attaches an existing .mdf and .idf file (containing Content Center library data) to the server. Use the Attach command to:

- Reattach to the server a library you previously detached.
- Attach a library (pair of .mdf and .idf files)

NOTE For information on the location of the libraries see the *Autodesk Data Management Server (version) Implementation Guide*. The guide is installed as a PDF in Program Files ► Autodesk ► ADMS [version] ► Help.

Use the Library Name dialog box to specify the library name, which may be different than the name of the .mdf file.

Import Copies library files (a pair of .mdf and .idf files) that are not in the database to the server console.

NOTE For information on the location of the libraries see the *Autodesk Data Management Server (version) Implementation Guide*. The guide is installed as a PDF in Program Files ► Autodesk ► ADMS [version] ► Help.

Context Menu for a selected library

Detach Removes Content Center libraries from the list of available libraries in the Content Center Library Manager dialog box. Detach does not delete them from the database. The libraries can be reattached and made available using the Attach command, or can be manually copied to another location, for example, to be backed up or moved to another computer.

Delete Detaches and deletes the selected library from the database. All data contained in the library is permanently deleted.

Export Copies a set of library files that are in the server console to a specified location. You can specify a folder on the same computer or a location available on the network.

Read Only Changes the status of a user-created read/write Library to read-only.

NOTE The read-only status of Autodesk Inventor standard and Parker libraries cannot be changed.

Copy	Copies a library from the Autodesk Inventor 2008 or 2009 partition to the Autodesk Inventor 2011 partition.
Edit Display Name	Edits the display name of the library.

NOTE For information on the location of the libraries see the *Autodesk Data Management Server (version) Implementation Guide*. The guide is installed as a PDF in Program Files ► Autodesk ► ADMS [version] ► Help.

Additional resources

32-bit operating systems: Setting the 3-GB switch

- 1 In Autodesk Inventor, click Help.
- 2 Select the Contents tab, and navigate to: Autodesk Inventor ► System Performance.
- 3 Double-click Increase Performance and Capacity, and then click the Procedure tab
- 4 Select Autodesk Inventor 3-GB feature: [Increase performance and capacity](#) on page 32

Service Packs and Hot Fixes: www.autodesk.com/inventor-updates-detail


Autodesk Labs: <http://labs.autodesk.com/>

Manufacturing Community: <http://mfgcommunity.autodesk.com/>

Discussion Groups: <http://discussion.autodesk.com/>

Graphics and Hardware: <http://www.autodesk.com/inventor-graphic-cards>

Inventor Help: In Autodesk Inventor, click Help.

-  On the Help home page, click 'Online Help - The Inside Track' to take a tour of the Help system.
- Providing feedback: Use the feedback link located at the bottom of every Help topic to provide immediate communication about the Help content. The comment links give you the opportunity to provide input into what you want and need from the Autodesk Manufacturing Technical

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IMPORTANT Remember to click **Back** to return to the CAD Manager Guide if you click any of the links on this page.

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