Cadmatic

Cadmatic eShare

Administration Guide | 2026H2

docs.cadmatic.com

Copyright © 2026 by Cadmatic

All rights reserved. No part of this publication may be reproduced or transmitted in any form or by any means without the express written permission of the publisher.

Cadmatic is a registered trademark of Cadmatic Oy. All other marks are property of their respective owners.

Cadmatic
Linnankatu 52
20100 Turku
Finland
Tel. +358 2 412 4500
www.cadmatic.com

Contents

1. Introduction 7
2. Integration with design applications 7
3. Integrated security and authentication 8
3.1. IIS requirements 8
3.2. Login session length 9
3.3. Legacy login 9
3.4. Related application settings in IIS Manager 9
4. OIDC provider support 9
4.1. Required application settings 10
5. Supported file formats 10
6. Getting started 13
7. Project administration 14
7.1. Opening a project 15
7.2. General 17
7.2.1. Editing project details 20
7.2.2. Exporting project configuration 21
7.2.3. Importing project configuration 21
7.2.4. Viewing the indexing status 22
7.3. Model configuration 23
7.3.1. Model upload and publishing 24
7.3.2. Creating an attribute configuration file 38
7.3.3. Attribute settings 39
7.3.4. Attribute categorization 59
7.3.5. Attribute visibility and order 64
7.3.6. Model hierarchies 65
7.3.7. Multiselect calculations 69
7.3.8. eGo QR configuration 72
7.4. Submodels 74
7.4.1. Configuring full model permissions 75
7.4.2. Configuring submodel division rules 76
7.5. Maps 81
7.5.1. Managing the visibility of a map to user groups 84
7.5.2. Replacing an existing map 85
7.6. Smart Point and markup types 85

7.6.1. Exporting and importing the Smart Point and markup type configuration 86
7.6.2. Importing Smart Points from Excel 87
7.6.3. Attribute types 89
7.6.4. Smart Point types 90
7.6.5. Markup types 93
7.7. Point clouds and textured meshes 96
7.7.1. Point clouds and textured meshes folder 97
7.7.2. Uploading point clouds 98
7.7.3. Verifying the display of point clouds 99
7.7.4. Creating zipped files of textured mesh files 100
7.7.5. Uploading textured meshes 100
7.7.6. Verifying the display of textured meshes 101
7.7.7. Managing the visibility of point clouds and textured meshes to user groups 101
7.7.8. Viewing point cloud and textured mesh file details 102
7.7.9. Managing point cloud and textured mesh hierarchies 102
7.7.10. Moving point clouds and textured meshes in folders 103
7.7.11. Removing point clouds or textured meshes 103
7.7.12. Exporting and importing point cloud and textured meshes configuration 104
7.7.13. AI label detection for point clouds 104
7.7.14. eShare point cloud converter 109
7.8. Status tracking and object grouping 110
7.8.1. Creating a new status tracker 112
7.8.2. Modifying existing status trackers 118
7.8.3. Exporting and importing status trackers 118
7.9. Adapters and data sources 118
7.9.1. Using attributes and tags in adapters 119
7.9.2. Adapters and indexing 120
7.9.3. Exporting and importing adapter configurations 121
7.9.4. Database adapter 122
7.9.5. Excel adapter 161
7.9.6. File system document adapter 173
7.9.7. Hyperlink adapter 177
7.9.8. M-Files adapter 183
7.9.9. Neo4j adapter 188
7.9.10. PI adapter 190
7.9.11. REST API adapter 192
7.9.12. SP Foundation adapter 214
7.10. Managed documents 217

7.10.1. Document hierarchies ... 217
7.10.2. User groups ... 219
7.10.3. Link indexing and document revisions ... 220
7.11. Document handling ... 223
7.11.1. Link highlighting ... 223
7.11.2. Document handlers ... 227
7.11.3. Document conversion formats ... 237
7.11.4. Linking multiline text strings ... 238
7.11.5. OCR configuration (beta version) ... 239
7.11.6. Exporting and importing document handling settings ... 241
7.12. Project users and groups ... 242
7.12.1. Adding and removing project users and groups ... 242
7.12.2. Exporting all project users and groups ... 245
7.12.3. Importing project users and groups ... 245
7.13. Project colors ... 246
7.14. Project variables ... 247
7.14.1. Setting project variables ... 247
7.14.2. Setting AI context ... 249
7.14.3. Setting permissions for AI data sharing ... 249
8. System administration ... 249
8.1. Dashboard ... 250
8.1.1. Viewing the indexing status of document data sources and managed documents ... 250
8.1.2. Configuring logging settings ... 251
8.1.3. Viewing and downloading the log ... 253
8.1.4. Changing log settings ... 254
8.2. Projects ... 254
8.2.1. Project management view ... 254
8.2.2. Adding a new project ... 256
8.2.3. Deleting a project ... 258
8.3. Users ... 259
8.3.1. User management view ... 259
8.3.2. Adding a new user ... 261
8.3.3. Editing user details ... 262
8.3.4. User activation and deactivation ... 263
8.3.5. Deleting a user ... 265
8.4. Groups ... 266
8.4.1. Group management view ... 266
8.5. Devices ... 269

8.5.1. Adding an activation key 269
8.5.2. Editing an activation key 270
8.5.3. Removing an activation key 270
8.5.4. Renewing a license 270
8.5.5. Removing a device 270
8.5.6. Exporting a device list to Excel 271
8.5.7. Importing a device list from Excel 271
8.6. Licenses 272
8.6.1. Adding a new user license group 273
8.6.2. Editing license group details 273
8.6.3. Deleting a user license group 273
8.7. JSON Web Tokens 274
8.7.1. Generating a JSON Web Token 274
8.8. External services 274
8.8.1. AI support 275
8.8.2. Map support 276
9. Command-line interface for administrative tasks 278
9.1. Auth JSON file format 281
10. eShare Backup Restore tool 281
10.1. Project backup 282
10.2. Server backup 283
10.3. Restoring eShare project from backup 284
10.3.1. Mapping users 285
10.4. Restoring eShare server from backup 286
10.5. Required permissions for eShare Backup Restore tool 288
10.5.1. SQL Server permissions 288
10.5.2. Windows / command-line prerequisites 289

1. Introduction

This document describes administrative tasks that a system administrator or project administrator can perform in Cadmatic eShare. If a task requires system administrator permissions, it is explicitly mentioned in the prerequisites of the task.

2. Integration with design applications

The Cadmatic design applications for plant and marine industries can be configured to connect to and exchange data with a Cadmatic eShare server. This integration enables project data created in design applications to be transferred directly to Cadmatic eShare, and 3D models and project documents to be published and made available to stakeholders who cannot access the design applications themselves.

In Cadmatic eShare, documents that are published from a Cadmatic design application are called *managed documents*, meaning that their content and linking with the 3D model is defined in the design application, and no further configuration needs to be performed in Cadmatic eShare. In addition to managed documents, Cadmatic eShare can also provide access to documents obtained from other sources, and these are called *external documents*.

All standard project administration functions of Cadmatic eShare can be applied to projects created in either Cadmatic eShare or a design application: you can define Smart Point Types, create new attributes, and add adapters to integrate projects with various external systems.

You must add the user account that the publishing service uses to the user list of Cadmatic eShare. For example, to allow Plant Modeller Service to publish to Cadmatic eShare, do the following:

- If Plant Modeller Service uses the Network Service account, and Plant Modeller and Cadmatic eShare are installed in the same computer, create the Cadmatic eShare user *NT AUTHORITY\NETWORK SERVICE*.
- If Plant Modeller Service uses the Network Service account, and Plant Modeller and Cadmatic eShare are installed in different computers, create the Cadmatic eShare user in the format DOMAIN\COMPUTERNAME$. For example, if domain is ACME and computername is PC-1, add the user as *ACME\PC-1$*.

- If Plant Modeller Service uses a named user account, add that user to the Cadmatic eShare user list, in the format DOMAIN\USERNAME.
- If Plant Modeller Service uses a local Windows account, create the Cadmatic eShare user in the format COMPUTERNAME\USERNAME.

Note that the NETWORK SERVICE name may be localized. Check the current Windows configuration to verify the correct name of NETWORK SERVICE if NETWORK SERVICE is used to run the service.

Note: If non-default font is used in managed documents, the corresponding font should be installed for All users in eShare server.

For more details on integrating a design application with Cadmatic eShare, and on working with models and documents created in a specific design application, refer to the online help of the application in question.

Related topics

User management view

3. Integrated security and authentication

The /Auth website and the eShareAuthPool application pool have been removed. Integrated security (Windows authentication) is now supported only on the /Login and /Login/Legacy endpoints. A Windows authentication login now issues a cookie, just like every other login type.

These changes are usually not visible to users, but they may be relevant if you troubleshoot login issues or maintain a customized IIS setup.

3.1. IIS requirements

Anonymous authentication must be enabled in Internet Information Services (IIS) Manager for eShare to function correctly.

eShare uses URL Rewrite to set the WWW-Authenticate response header, so this server variable must be allowed in *applicationHost.config*:

```
<rewrite>
    <allowedServerVariables>
        <add name="RESPONSE_WWW_Authenticate" />
    </allowedServerVariables>
</rewrite>
```

The eShare installer adds this automatically. If you configure IIS manually, or the setting has been removed, add it yourself.

3.2. Login session length

For OIDC and Microsoft Entra ID logins, the cookie expires when the original token expires. For Windows authentication, no token expiration is available, so the cookie uses the default expiration of 12 hours. You can change this with *Cadmatic.Auth.Cookie.DefaultExpirationSeconds*. After upgrading, users may need to log in more often than before.

To use the default expiration for all logins, regardless of how long the original token is valid, set *Cadmatic.Auth.Cookie.ForceDefaultExpiration* to true.

3.3. Legacy login

Earlier versions of Cadmatic tools use the /Login/Legacy endpoint. It is enabled by default during the migration period and will be disabled by default in a future version. If older Cadmatic tools cannot log in to eShare, check that *Cadmatic.Auth.IsLegacyLoginEnabled* is set to true.

3.4. Related application settings in IIS Manager

- *Cadmatic.Auth.IsLegacyLoginEnabled*
- *Cadmatic.Auth.Cookie.DefaultExpirationSeconds*
- *Cadmatic.Auth.Cookie.ForceDefaultExpiration*

For more information, see Configuring application settings.

4. OIDC provider support

Cadmatic eShare has custom OIDC provider support. This enables using third-party and in-house ID providers instead of Windows or Entra ID/Azure AD users.

Note: You cannot configure OIDC in the eShare installer.

Prerequisites

- An existing OIDC provider

You need to set up the following to use OIDC with eShare:

- Configuration to set up OIDC to work with eShare
- Application settings configured to use OIDC. See Required application settings

4.1. Required application settings

Specify the following settings in the application settings in Internet Information Services (IIS) Manager:

- *Cadmatic.OIDC.AudienceList* (optional)
- *Cadmatic.OIDC.Authority*
- *Cadmatic.OIDC.ClientId*
- *Cadmatic.OIDC.ClientSecret* (optional)
- *Cadmatic.OIDC.LoginClaim*
- *Cadmatic.OIDC.ResponseType*
- *Cadmatic.OIDCSecurity*

For more information, see Configuring application settings.

When setting up OIDC, use */Oidc/signin-oidc* as the eShare sign-in redirect endpoint, and */Oidc/signout-callback-oidc* as the sign-out redirect endpoint.

For third-party integrations, it is possible to call eShare public APIs directly with OIDC access tokens as a bearer token, as long as eShare client ID is one of the token's audiences.

5. Supported file formats

Cadmatic eShare enables combining 3D models from various file formats. Point cloud files are also supported, along with textured meshes.

The table below lists software systems supported by Cadmatic eShare and their associated file formats.

Company	Software	Files
Aveva	PDMS You can import files to Cadmatic eShare using E3D, PDMS and 3DD model importer.	.rvm, .att, .3dd

Company	Software	Files
	The RVM file should be in text format. For more information, see Model upload and publishing.	
Autodesk	**AutoCAD** You can import files to Cadmatic eShare using DWG/DWF/DWFx/DXF/DGN model importer. For more information, see Model upload and publishing.	.dwg, .dxf, .dwf, .dwfx
	FBX You can upload textured meshes to Cadmatic eShare as FBX files with an embedded texture image in PNG or JPEG format, or the texture linked as a separate file in PNG or JPEG format. For more information, see Point clouds and textured meshes.	.fbx
	Navisworks You can import files to Cadmatic eShare using Navisworks model importer. For more information, see Model upload and publishing.	.nwc, .nwd
Bentley	**MicroStation** You can import files to Cadmatic eShare using DWG/DWF/DWFx/DXF/DGN and 3DD model importer. For more information, see Model upload and publishing.	.dgn, .3dd
	PTX Point cloud For more information, see Point clouds and textured meshes.	.ptx
buildingSMART	**Industry Foundation Classes** You can import files to Cadmatic eShare using IFC model importer. For more information, see Model upload and publishing.	.ifc, .ifczip
Cadmatic	**Cadmatic models**	.ebm,

Company	Software	Files
	You can import files to Cadmatic eShare using EBM/EBMX and 3DD model importer. For more information, see Model upload and publishing.	.ebmx, .3dd
	Cadmatic point cloud files You can upload point cloud files to Cadmatic eShare. For more information, see Point clouds and textured meshes.	.cpd, .cpe, .cpx
Hexagon PPM	**Intergraph PDS** You can import files to Cadmatic eShare using PDS and 3DD model importer. For more information, see Model upload and publishing.	.dgn, .3dd
	Intergraph Smart 3D (SmartPlant) You can import files to Cadmatic eShare using SmartPlant model importer. For more information, see Model upload and publishing.	.vue, .mdb2
	Leica You can upload point cloud files to Cadmatic eShare. For more information, see Point clouds and textured meshes.	.ptx
The Khronos Group	**GLB** You can upload textured meshes to Cadmatic eShare as GLB files with an embedded texture image in PNG or JPEG format. For more information, see Point clouds and textured meshes.	.glb
Open Design Alliance	**Industry Foundation Classes** You can import files to Cadmatic eShare using IFC model importer. For more information, see Model upload and publishing.	.ifc
Siemens	**Siemens NX**	.jt

Company	Software	Files
	You can import files to Cadmatic eShare using JT model importer. For more information, see Model upload and publishing.	
Trimble	**Tekla Structures** You can import files to Cadmatic eShare using IFC model importer. For more information, see Model upload and publishing.	.ifc, .ifczip
Wavefront Technologies	**OBJ** You can upload textured meshes to Cadmatic eShare as OBJ files with the texture linked as a separate image in PNG or JPEG format. For more information, see Point clouds and textured meshes.	.obj
	e57 Point cloud You can upload point cloud files to Cadmatic eShare. E57 files must contain either structured point cloud with scanner position information or unstructured point cloud with spherical panorama images. For more information, see Point clouds and textured meshes.	.e57

6. Getting started

After installing Cadmatic eShare, the first user who logs in is granted system administrator permissions, and this first user can then add other users, either as system administrators or normal users. All users access eShare using either a web browser or the eShare App.

If eShare is running on the local computer in port 81, you can access the system by opening *https://localhost/* in a web browser and signing in with your Microsoft Windows credentials if requested to do so. Login information is never transmitted between the web browser and the server. Login requires a license to be available, and if all licenses are already in use, an administrator level user can select to take the license from another user.

- To open system administration, select **System Admin** in the menu.



In the System Administration view a system administrator can do the following:

- Manage projects as described in Project management view.
- Manage users as described in User management view.
- Manage user groups as described in Group management view.
- Manages licenses as described in Licenses.
- Manage devices as described in Devices.
- Manage AI support as described inAI support.

7. Project administration

In Cadmatic eShare, a project is a collection of data that relates to plant or ship design and can be accessed with a web browser. Typically, this data consists of a 3D model obtained from a CAD system and any number of drawings and other documents obtained from Cadmatic design applications or external data sources such as document management systems.

System administrators can access and manage all projects in the system. They can create and delete projects, and they can edit, export, and import project configurations. See System administration.

Normal users can access a project after they have been assigned to that project, either as a project administrator or project user.

- Click **Project Admin** in the main menu to open the project administration view.



7.1. Opening a project 15
7.2. General 17
7.3. Model configuration 23
7.4. Submodels 74
7.5. Maps 81
7.6. Smart Point and markup types 85
7.7. Point clouds and textured meshes 96
7.8. Status tracking and object grouping 110

7.9. Adapters and data sources 118
7.10. Managed documents 217
7.11. Document handling 223
7.12. Project users and groups 242
7.13. Project colors 246
7.14. Project variables 247

7.1. Opening a project

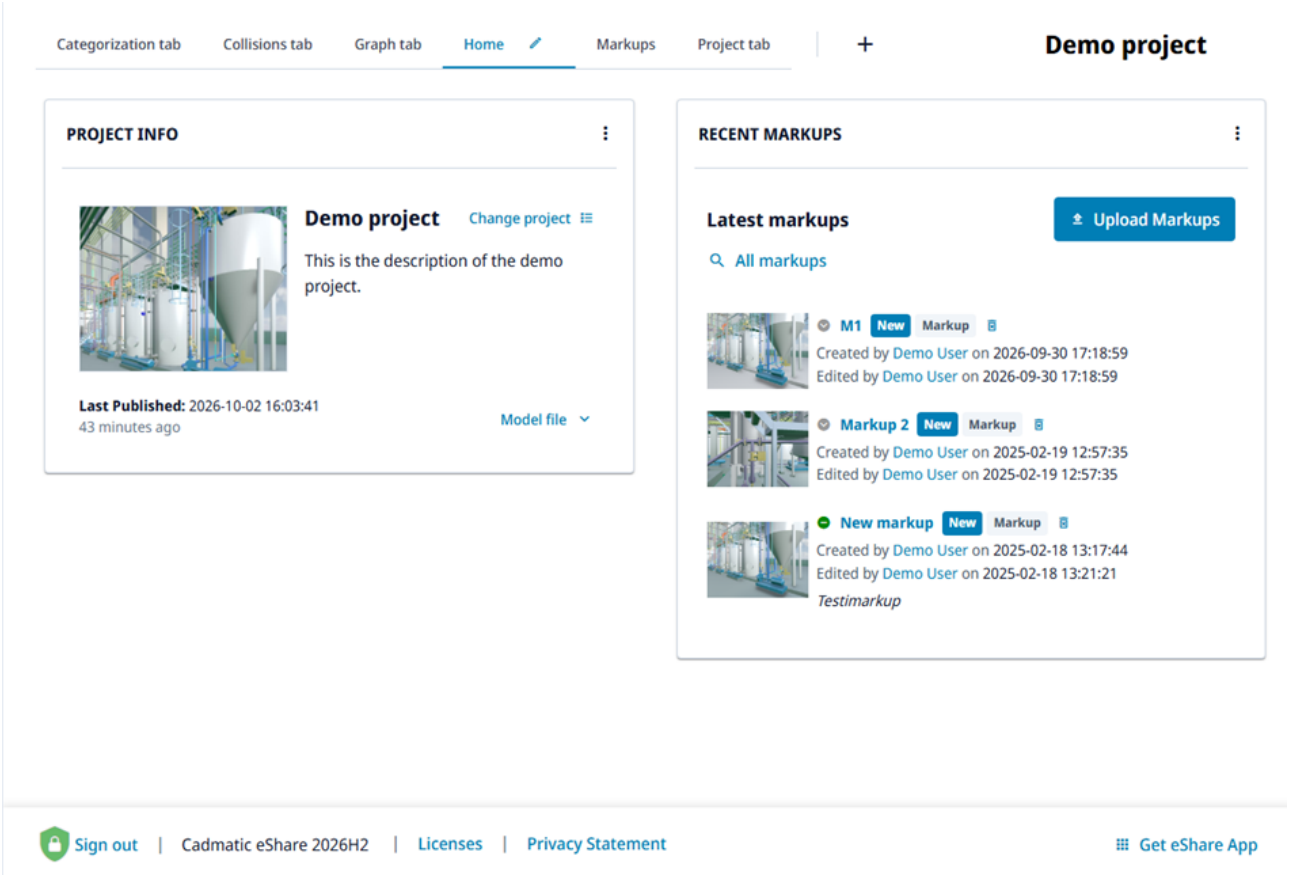
When a new user opens Cadmatic eShare for the first time, the main toolbar displays "No Project" to indicate that this user has not opened any projects yet. When a project is already open, instead of "No Project" the main toolbar displays the name of the open project.

Clicking "No Project" or the displayed project name opens the **Projects** list. The list shows all projects that this user is permitted to access by being assigned to the project either as a project administrator or normal project user.

Projects can also be selected from the map using  button to open the map view. If the button is not visible, system administrator has not enabled map support. See Map support.

You can filter the project list by typing a part of the project's name in the **Filter** field. You can also open the filtered project list from a URL that has the format https://<server>/#/selectProject?filter=<projectTitle>. For example, *https://localhost/#/selectProject?filter=example*.

To open a project, click the project name on the **Projects** list. The project front page opens.



Depending on user level—system administrator, project administrator, or normal user—and current project configuration, the project front page can contain some or all of the following elements:

- Model file date. Clicking the arrow shows the dates of when the model was last published, uploaded, and modified. Clicking **Download Model** downloads the model file in EBMX format.
- The download link can be enabled or disabled in Internet Information Services (IIS) Manager application settings and permissions in **Project Admin > Submodels**. See Configuring Application Settings and Submodels.
- Project name.
- If the project administrator has enabled it, **Upload Markups** button opens a file browser for importing a Cadmatic Markup file (*.ebx) or a BCF file to the project.
- You can configure in the recent markups widget, if the mode is to show markups for the current user or for all users.



- **My latest markups** lists the five latest Markups that you have added to the model. You can click **All Created by Me** or **All Modified by Me** to see more Markups you have added or edited in this project.
- **Latest markups** lists the five latest Markups that any user has added to the model. You can click **All Markups** to see more Markups.

7.2. General

Clicking **General** in the menu opens the project configuration view where project administrators can configure the project in the following views.

Configuration View	Description
General	Displays project name, database name, project image, project description, and the name of the template project if one was used when creating the project. Displays the number of projects currently publishing, status of point cloud synchronization, and the indexing status of document data sources and managed documents. The name of the data source is a link to the data source configuration. See Viewing the indexing status . **Note:** If you edit and save the data source configuration, all data sources of the same adapter with indexing enabled, will be indexed again. The project details can be edited. See Editing project details. Complete project settings be exported and imported. See Exporting project configuration and Importing project configuration. Database Name displays the project ID. The ID is needed when adding or removing point cloud files (.cpd). Point cloud files are stored in the eShare server in

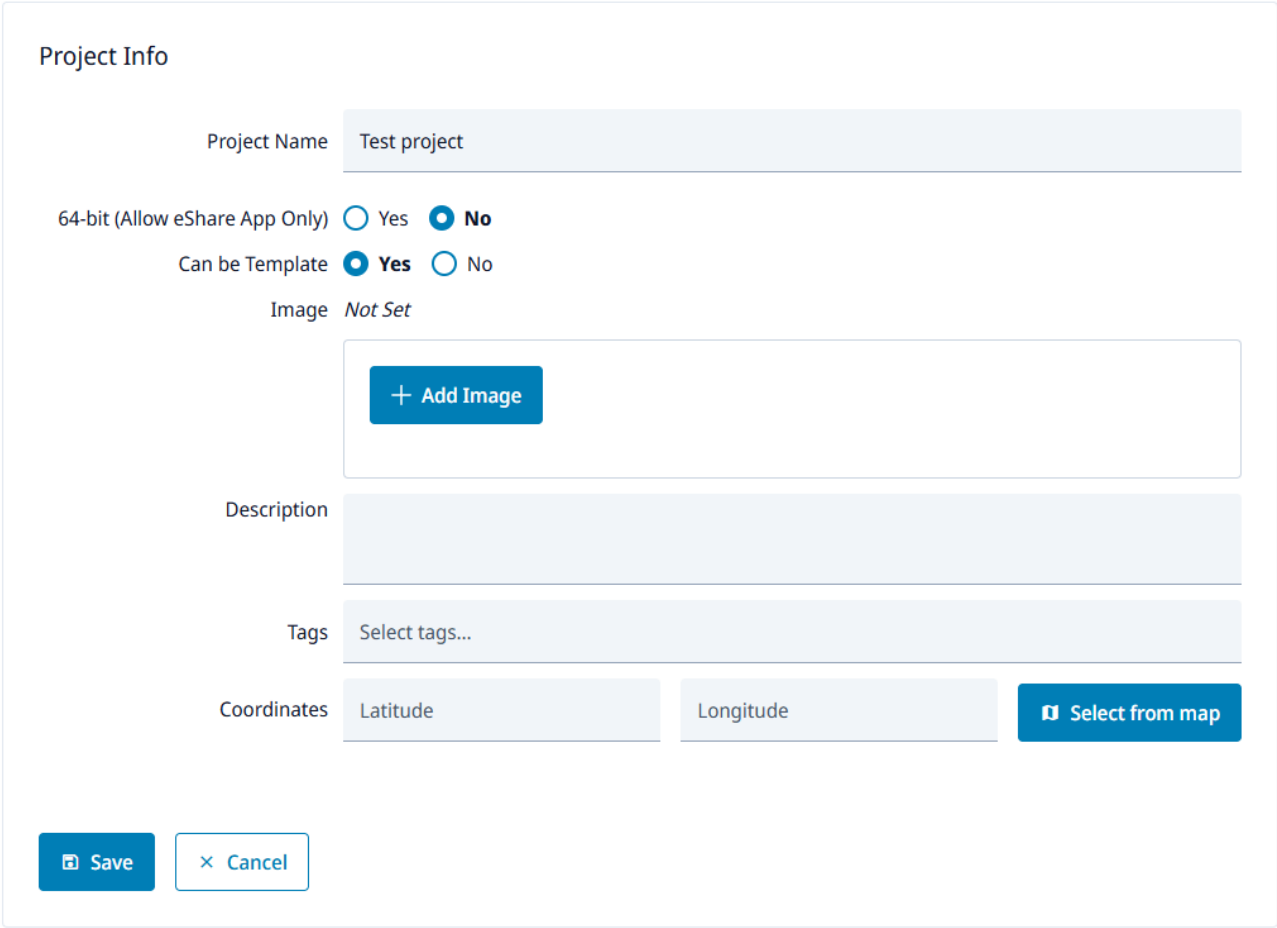
Configuration View	Description
	C:\ProgramData\Cadmatic\eShare\pointCloud\<project id>.
Model configuration	Allows configuring attribute settings, attribute categorization, and attribute visibility, as well as uploading and publishing a 3D model. See Model configuration.
Submodels	Allows configuring settings for publishing submodels. See Submodels.
Maps	Allows uploading 2D maps of the model. See Maps.
Smart Point and Markup Types	Allows configuring Smart Point Types, Markup Types and Attribute Types. See Smart Point and markup types.
Point Clouds and Textured Meshes	Allows uploading and removing point clouds and textured meshes, view their status and information, creating hierarchies, and downloading generated point cloud files. See Point clouds and textured meshes.
Status Tracking and Object Grouping	Allows configuring status tracking definitions and designating status workflows that define which statuses can each status be changed to. See Status tracking and object grouping.
Adapters and Data Sources	Allows integrating Cadmatic eShare with other systems and applications, to retrieve and display information for example from a relational database, a document management system, a Microsoft Excel file, or web page. See Adapters and data sources.
Managed Documents	Allows specifying the hierarchy of documents obtained from Cadmatic design applications as well as designating how object links are highlighted in such documents. See Managed documents.
Document Handling	Allows configuring automatic detection of references to design objects in documents obtained from an external data source or Cadmatic design application. See Document handling.
Project Users and Groups	Allows adding and removing project users, and designating users as project administrators. See Project users and groups.

Configuration View	Description
Project Colors	Allows defining a color palette for the project to be used in all color pickers. See Project colors.
Project Variables	Allows creating variables that can be used in adapter and data source configurations. Editing a project variable that is already in use requires opening and saving each configuration where the variable is being used. See Project variables.

All of the above—the complete project configuration—can be transferred from one project to another by using project configuration export and import.

7.2.1. Editing project details

You can edit project details in project administration view.



Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. In the **General** view, click **Edit** in Project Info pane. The **Project Info** view opens.
3. Edit the project information.

   You can add project tags by clicking the cursor in the field and select from existing tags or enter a new one. To remove a tag, select ⊗.

   Coordinates can be entered manually or selected from the map using **Select from map**, if Map support is enabled for the project by the system administrator. See Map support.

The first time the map is opened, it prompts for location permission. If enabled, it will initialize the location at your current location, or if the project has a location already that will be used instead. Otherwise a predetermined location is used. You can select a location from the map by clicking, which updates the coordinates and is visualized with a marker on the map. To close the map, select **Close map**.

4. Click **Save**.

7.2.2. Exporting project configuration

You can export the complete configuration of a project in text format. You can use this to easily copy settings from one project to another, or just to back up project settings. The export does not include the 3D model or any documents.

Note: The same settings are copied when you create a new project and select to use a Template Project as described in Adding a new project. Many administrative views also allow you to export or import just those settings that are specific to that view.

The export preserves variables—for example, if the source project uses a *{projectName}* or *{projectDbName}* variable as part of a data source configuration or in the connection string of a database adapter, in the export it will not be replaced with the actual project name or project database name.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. In the **General** view, click **Export Config**. The **Export Project** text box displays the configuration in text format.
3. Copy the contents of the text box to the clipboard. You can save the copied settings in a text file if needed.

Related topics

Importing project configuration

7.2.3. Importing project configuration

You can import a previously exported project configuration to a project, for example to use consistent settings across all projects, or to restore original project settings from a backup. Entities

that already exist are skipped or updated, or the imported entity is renamed with a running number, depending on entity type.

The import does not affect 3D models or documents that might exist in the target project.

Prerequisites

- You have a project configuration export on the clipboard. See Exporting project configuration.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. In the **General** view, click **Import Config**. The **Import Project** text box is displayed.
3. Paste the configuration to import into the text box.
4. **Import project administrator permissions?** – Select whether to import project administrator permissions.
    - **Yes** – Users that have project administrator status in the imported configuration will also have project administrator status in the target project.
    - **No** – All users are imported as normal project users.
5. **Import default markup properties?** – Select whether to import the properties of the default Markup Type (User Groups, Model Position, Assignee).
    - **Yes** – Import the properties of the default Markup Type.
    - **No** – Keep the current properties of the default Markup Type.
6. Click **Import**. The program reports if any entities were skipped or renamed in import.

7.2.4. Viewing the indexing status

Do the following:

1. Click **Project Admin** in the main menu. General view opens.
2. Scroll down on the view until publishing and document indexing data is shown.
3. See **Document indexing** section for the indexing status of document data sources and managed documents.
4. You can make the following selections:
    - Click the name of the data source to see its data source configuration. **InternalDocuments** are managed documents.

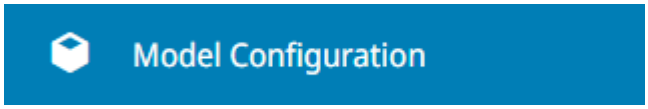
Note: If you save the data source configuration, all data sources of the same adapter with indexing enabled, will be indexed again.

- To clear cached document data, click . This will delete the cached data and the text extraction will be ran again. This can be useful in cases where the text extraction has failed and you want to try again. The text extraction should run every time the document is updated, but this can also be forced by clicking the button.
- To clear the indexing results, click . This will remove all the indexing results from the database, nullifying all indexing of the data source.
- To restart the indexing, click . The indexing will be restarted and completed using the indexing speed set in the data source configuration.
- To restart the indexing and complete it as fast as possible, click . The indexing will be restarted and completed as fast as possible by forcing the data source to use As Fast as Possible as the indexing speed.

7.3. Model configuration

In the model configuration view, project administrator can publish 3D models, manage object attributes, define rules for submodel publishing, edit and add model hierarchies, and enable the QR code reader of Cadmatic eGo to read custom codes.

- Click **Model Configuration** to open model configuration view in project administration.



Models can be added to Cadmatic eShare in three ways:

- Cadmatic Plant/Outfitting can send the 3D model of a design project directly to eShare.
- Cadmatic Plant/Outfitting and Cadmatic Hull can export the 3D model as an .ebm/.ebmx file, and project administrator can upload the .ebm/.ebmx file to eShare.
- Project administrator can import 3D models in multiple formats directly to eShare.

After a model has been added to eShare, project administrator can publish the model, and then the model will be visible to users.

Note: Models that have an expiry date cannot be used in eShare.

Attributes describe the properties of the graphical objects that comprise the 3D model: their dimensions, mass, and position ID, for example. In the model configuration view, project administrator can specify which attributes to show to the users, define a separate display name for each attribute, and even generate new attributes by extracting data from existing attributes. Attributes can be used in various ways to retrieve information about the model objects and visualize this information to the users. In the 3D viewer, the available attributes are displayed when the user selects to examine an object. Users can also search for objects that have specific attributes or attribute values, and the 3D viewer can show objects in different colors based on their attributes.

Project administrator can also define rules for dividing the model into submodels, which contain only specific objects instead of the full model. Submodels are useful when the full model is too large to open or download efficiently, or when users only need a specific part of the model. Access to the full model and to each submodel can be restricted by user group. See Submodels.

7.3.1. Model upload and publishing

In the model configuration view, project administrator can upload 3D model files and publish models to the project users. Multiple models can be published together by using coordinate transformations. All model importers have general configuration (including translation and attribute renaming). Some model importers, for example JT, have importer specific settings as well.

EBMX models published with Cadmatic Plant Modeller 2026H2 are not compatible with older eShare versions. eShare should be updated to 2026H2 to support all file versions.

If you have projects created in 2023T2 containing submodels, they will need to be re-published.

7.3.1.1. Obtaining model files from design tools

Cadmatic eShare can use 3D models created with various design tools. The supported formats are EBM, EBMX, 3DD, DGN, DWG, DXF, DWF, DWFx, IFC, IFCzip, JT, NWC, NWD, PDMS, PDS, and SmartPlant.

Other Cadmatic applications can publish/export .ebm/.ebmx files as follows:

- Plant Modeller: **Model > eBrowser Model > Publish**. (Models must be published without an expiry date.)
- Hull Viewer: **File > Save As > eBrowser Model (.ebm)**

You can merge several EBM files together by using a licensed version of Cadmatic eShare or Cadmatic eBrowser.

7.3.1.2. Using model importers

Model importers are needed for importing 3D models from different formats to eShare. When a new project is created, an EBM/EBMX model importer is included in the project by default. The default model importer is used for importing models published directly from Plant Modeller. The default model importer can be edited or disabled but not removed.

Note: Model importers for JT, PDMS, PDS, and SmartPlant need their own separate licenses.

7.3.1.2.1. Using E3D eXchanger

Prerequisites

An administrator needs to first create a configuration for the importer using the desktop E3D eXchanger and save the project as a json file. The input rvm and att files are added in a zip file with the project.json, which is then uploaded to eShare. The E3D importer in eShare requires a **Configuration file**, which is the name of the project.json in the uploaded zip file. If the field is left empty, eShare will try to use the first .json file it finds in the zip file.

Note: The uploaded zip file needs to have all the files referenced in the configuration file in the root of the zip file.

7.3.1.2.2. Using PDMS/PDS eXchanger

PDMS eXchanger is now deprecated, use E3D eXchanger instead.

Prerequisites

- Before importing PDMS/PDS models, an administrator must install a separate PDMS/PDS eXchanger locally in the same server where eShare server is running.
- Either *Cadmatic.PdmsExchangerFolderName* for PDMS, or *Cadmatic.PdsExchangerFolderName* for PDS setting in Internet Information Services (IIS) Manager must match the installation folder of the PDMS/PDS eXchanger. See Configuring application settings.

7.3.1.2.3. Creating a model importer

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. To add a new model importer, click **Add Model Importer**.
4. Select the correct type of model importer.
5. Click **Create Importer**.
6. Enter the following information.

   Model Importer

   - **Name** – Enter a name for the importer. The name can only contain the following characters: *a-zA-Z0-9_-()space*
   - **Description** – Enter a description for the importer.
   - **State** – Select if the importer is enabled or disabled.

   Attributes Configuration

   - **Abbreviation Renaming** – If set to **Manual**, the attributes are retained as they are in the model. Identical attributes will be shared between models, but they can be renamed manually.

     In **Abbreviation Processing Rules** you can add rules for renaming abbreviations in **Rename**, creating new attributes in **Create**, or deleting attributes in **Delete**.

     Note: If a new attribute is added, it is applied globally to all objects during model import.

     Note: The new attributes only take effect after the model is published and the model is reloaded.

     If Abbreviation renaming is set to **Automatic**, the model will not share attribute names with other models. You can define a unique suffix for the attributes of the model in the **Custom Suffix** field or leave it empty to use the file name as an identifier.

   In **Transform / Move**, **Transform / Scale** and **Transform / Rotate** set the coordinate transformations if needed.

   Note: Move, rotate, or scale cannot be used if the model defines local coordinate systems.

Import Configuration (only used when uploading)

- If you are creating a DWG/DWF/DWFx/DXF/DGN model importer, select the following:
  - **Transformation** – Specify the following:
    - **Transformation Type** – If you select **None**, the model is not transformed during import. If you select **Translation**, the model is moved, and the coordinates change during import. Set the X, Y, and Z values. If you select **Local origin**, the origin for the model is set, allowing large coordinates to be handled. The model is shown with the same coordinates as in the original drawing, but the internal storing of the coordinates is changed during import.

      This setting should be used, if the model is in the desired location and the coordinates should be shown similarly as in the original drawing, but the model is more than 10 km from the origin.
  - **Cut Box** – Specify the following:
    - **Exclude geometry outside cut box**– If you select **Yes**, a cut box can be defined with minimum and maximum points for the model area. By using a cut box you can cut out geometry that is incorrectly modeled and could cause issues, or is outside the project area. Cut box only removes geometry that is completely outside of the defined cut box, and object data will remain in the model. Cut box should be defined in the same coordinates as the model is originally before import.
  - **Import Settings** – Specify the following:
    - **Include freeze layers** – If you select **Yes**, layers with freeze status are included in the imported model.
    - **Include off layers** – If you select **Yes**, layers with off status are included in the imported model.
    - **Include wireframes** – If you select **Yes**, plain wireframe objects are included in the imported model. If wireframes are not included, it can affect rendering performance, and will result in a smaller imported model.
  - **Optimization** – Specify the following:
    - **Split entities to retain materials** – If you select **Yes**, entities will be split to retain correct visualization material during import, and object count might increase. If you select **No**, some entities might get incorrect color.

        - **Geometry reduction** – If you select **Keep geometry as is**, the geometry is kept as it is in the original drawing. If you select **Remove duplicate vertices**, duplicate vertices are removed. If you select **Reduce triangles, target triangle count**, the number of triangles is reduced to the set target in **Reduction target triangle count**.
- If you are creating an E3D model importer, select the following:
    - **Importer configuration**
        - **Configuration file** – Enter the configuration file name, i.e. the name of the project.json file. If left empty, eShare will try to use the first .json file it finds in the zip file.
- If you are creating an EBM/EBMX model importer, select the following:
    - **Import Settings** – Specify the following:
        - **Merge Attributes with Same Abbreviation** – When the model has multiple attribute definitions with the same abbreviation but possibly different other properties, the two attributes are merged regardless, if set to **Yes**.
- If you are creating an IFC model importer, select the following:
    - **Transformation** – Specify the following:
        - **Transformation Type** – If you select **None**, the model is not transformed during import. If you select **Translation**, the model is moved, and the coordinates change during import. Set the X, Y, and Z values. If you select **Local origin**, the origin for the model is set, allowing large coordinates to be handled. The model is shown with the same coordinates as in the original drawing, but the internal storing of the coordinates is changed during import.

          This setting should be used, if the model is in the desired location and the coordinates should be shown similarly as in the original drawing, but the model is more than 10 km from the origin.
    - **Cut Box** – Specify the following:
        - **Exclude geometry outside cut box** – If you select **Yes**, a cut box can be defined with minimum and maximum points for the model area. By using a cut box you can cut out geometry that is incorrectly modeled and could cause issues, or is outside the project area. Cut box only removes geometry that is completely outside of the defined cut box, and object

data will remain in the model. Cut box should be defined in the same coordinates as the model is originally before import.

- **Import Settings** – Specify the following:
  - **Include Wireframes** – If you select **Yes**, plain wireframe objects are included in the imported model. If wireframes are not included, it can affect rendering performance, and will result in a smaller imported model.
  - **Include Parent Attributes** – If you select **Yes**, parent group attributes of objects are included in the imported model. If parent attributes are not included, the imported model will include only the attributes of objects.
  - **Include Empty Valued Attributes** – If you select **Yes**, attributes with empty values are included in the imported model.
  - **Include all representation contexts** – If you select **Yes**, all IFC representation contexts will be included in the imported model, and not only the representation contexts defined in the IfcProject.
- **Optimization** – Specify the following:
  - **Split IFCProducts to retain materials** – If you select **Yes**, the IFCProducts are split as several objects to preserve correct visualization materials for all parts. If enabled, object count in the imported model might increase.
  - **Geometry reduction** – If you select **Keep geometry as is**, the geometry is kept as it is in the original drawing. If you select **Remove duplicate vertices**, duplicate vertices are removed. If you select **Reduce triangles, target triangle count**, the number of triangles is reduced to the set target in **Reduction target triangle count**.
  - **Importer Threading**– If you select **Multithread**, the importer will run on multiple threads (faster import). If you select **Single Thread**, the importer will run on a single thread (safer import). The default is Multithread.

- If you are creating a JT model importer, select the following:
  - **Conversion Type** – LOD 0-9 specifies the level of detail. XT specifies conversion to use XT B-Rep elements.
- If you are creating a Navisworks model importer, select the following:
  - **Transformation** – Specify the following:
    - **Transformation Type** – If you select **None**, the model is not transformed during import. If you select **Translation**, the model is moved, and the

coordinates change during import. Set the X, Y, and Z values. If you select **Local origin**, the origin for the model is set, allowing large coordinates to be handled. The model is shown with the same coordinates as in Navisworks, but the internal storing of the coordinates is changed during import.

This setting should be used, if the model is in the desired location and the coordinates should be shown similarly as in Navisworks, but the model is more than 10 km from the origin.

- **Cut Box** – Specify the following:
  - **Exclude geometry outside cut box** – If you select **Yes**, a cut box can be defined with minimum and maximum points for the model area. By using a cut box you can cut out geometry that is incorrectly modeled and could cause issues, or is outside the project area. Cut box only removes geometry that is completely outside of the defined cut box, and object data will remain in the model. Cut box should be defined in the same coordinates as the model is originally before import.
- **Import Settings** – Specify the following:
  - **Group attributes** – Select one of the following:
    - **Ignore** – Group attributes are ignored.
    - **Collapse to objects** (default) – Group attributes are collapsed to objects.
    - **Include groups in import** – Group attributes are included in the imported model.
  - **Include Hidden Objects** – If you select **Yes**, objects with hidden state are included in the imported model.
  - **Include Wireframes** – If you select **Yes**, plain wireframe objects are included in the imported model. If wireframes are not included, it can affect rendering performance, and will result in a smaller imported model.
  - **Include Empty Valued Attributes** – If you select **Yes**, attributes with no value are included if necessary. By default they are skipped.
  - **Default Font** (optional) – Defines the fallback font used, if the Navisworks model font is not found in system.
- **Optimization** – Specify the following:

        - **Split objects to primitives sharing geometry** – If you select **Yes**, objects are split to primitives to reduce memory consumption of the imported model and to improve rendering performance.
        - **Geometry reduction** – If you select **Keep geometry as is**, the geometry is kept as it is in the original drawing. If you select **Remove duplicate vertices**, duplicate vertices are removed. If you select **Reduce triangles, target triangle count**, the number of triangles is reduced to the set target in **Reduction target triangle count**.
    - **Import Settings** – Specify the following:
        - **Merge Attributes with Same Abbreviation** – When the model has multiple attribute definitions with the same abbreviation but possibly different other properties, the two attributes are merged regardless, if set to **Yes**.
- If you are creating a PDMS or PDS model importer, specify the following:
    - **Project File** – Enter the name of the project file. The PRJ file and all the files referenced in the PRJ file must be in the uploaded .zip archive. For example in PDMS this usually means RVM (text format, binary is not supported), ATT files for the model data and attributeMapping, ColorSettings and other configuration files used in the import.

      If the file in the archive is located in a subfolder, add the name of the subfolder in the field. For example *data\Project.prj*.
- If you are creating a SmartPlant model importer, select the following:
    - **Geometry File** – Enter the name of the geometry file. The file must be in the uploaded .zip archive. If the file in the archive is located in a subfolder, add the name of the subfolder in the field. For example *data\Geometry.vue*.
    - **Attribute File** – Enter the name of the attribute file. The file must be in the uploaded .zip archive. If the file in the archive is located in a subfolder, add the name of the subfolder in the field. For example *data\MyAttributeFile.mdb2*.
    - **Attribute Filter File** – Enter the name of the attribute filter file. If no attribute filter file is used, leave the field empty. The file must be in the uploaded .zip archive. If the file in the archive is located in a subfolder, add the name of the subfolder in the field. For example *data\AttributeFilter.txt*.
    - **Model Unit** – Select the measurement unit used in the model.

- **Translate Origin** – If you select **Yes**, the model is moved, and the coordinates change during import. If you select **No**, the model is not transformed during import.

7. Click **Save**.

Note: You can add an attribute configuration file for DWG, IFC, and Navisworks model importers. See Adding an attribute configuration file for a model importer.

7.3.1.2.4. Disabling or enabling a model importer

A model importer can be disabled or enabled in the project.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Select the correct model importer.
   - To disable an enabled importer, select **Disable**.
   - To enable a disabled importer, select **Enable**.

7.3.1.2.5. Exporting a model importer

A model importer can be exported to be imported to another project.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Click the name of the model importer.
4. Click **Export**.
5. Copy the model importer strings from the field.
6. Click **Close**.

7.3.1.2.6. Exporting all model importers

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Click **Export All**.
4. Copy the model importer strings from the field.
5. Click **Close**.

7.3.1.2.7. Importing a model importer

An exported model importer can be imported to a project.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Click **Import**.
4. Paste the exported model importer strings to the field.
5. **Overwrite default importer** – If set to **No**, the importer will not overwrite the default importer. If set to **Yes**, the model importer will overwrite the default importer if the imported configuration contains a default importer.
6. Click **Import**.

7.3.1.2.8. Editing or removing a model importer

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Select the correct model importer.
    - To edit the importer, click on the edit button . Model importer edit view opens. Make changes and click **Save**. Click **Cancel** to undo changes.

- To remove the importer, click on the remove button . A confirmation dialog opens. Click **Remove** to confirm or **Cancel** to keep the importer.
  The default model importer cannot be removed.

7.3.1.2.9. Adding an attribute configuration file for a model importer

An attribute configuration file defines how attributes should be interpreted and imported from a model. An attribute configuration file (TXT format) can be added to DWG, IFC, and Navisworks model importers. After the attribute configuration file is added to the model importer, it is included in the import and export of the model configuration, and system backup and restore.

Prerequisites

- You have an attribute configuration file. See Creating an attribute configuration file.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Select the correct model importer.
4. In **Attribute configuration file**, select **Upload** to upload the attribute configuration file.
   Once the attribute configuration file is uploaded, the file name and upload time is visible.
5. Click **Publish** to publish the model. See Uploading and publishing a 3D model.
6. Open Model view and reload the model for changes to take effect.

7.3.1.2.10. Downloading an attribute configuration file

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Select the correct model importer.
4. In **Attribute configuration file**, select **Download** to download the attribute configuration file.

7.3.1.2.11. Deleting an attribute configuration file

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Select the correct model importer.
4. In **Attribute configuration file**, select **Clear** to remove the attribute configuration file.

7.3.1.3. Uploading and publishing a 3D model

Project administrator can import a 3D model from different file formats. If the project already contains a model, the imported model will replace the existing model when the uploaded model is published.

Note: Having submodels configured in the project can affect the publishing time as they are published as part of the publishing process.

Prerequisites

- Model does not have an expiry date. Models that Plant Modeller publishes with an expiry date cannot be uploaded to eShare.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Select the correct model importer. Click **Upload**, browse to the file to upload, and click **Open**. The model is uploaded to the server. Or click **Upload and Publish** to upload and publish the model consecutively.
4. Before publishing the model to users, you can edit various settings related to the attributes in the model. See the following topics for details.
    - Attribute settings
    - Attribute categorization
    - Attribute visibility and order
5. In Model Publishing section, specify the following settings:
    - **Keep models separate** – If you have multiple models uploaded to the project, all of the models will be published by default. To keep the models separate in the model tree, select **Yes**.

- **Add bounding box min coordinates** – Set to **Yes**, if you want to add the object bounding volume minimum coordinates to the published model.
- **Add bounding box max coordinates** – Set to **Yes**, if you want to add the object bounding volume maximum coordinates to the published model.
- **Add bounding box center coordinates** – Set to **Yes**, if you want to add the object bounding volume center coordinates to the published model.

Note: Attributes related to these selections are not visible in Attribute settings, unless they are enabled in the model configuration.

- **Merge attributes with same abbreviation** – When the model has multiple attribute definitions with the same abbreviation but possibly different other properties, the two attributes are merged regardless, if set to **Yes**.
- **Normalize attribute values** – When set to **Yes**, user-input in model attributes is cleaned up to ensure equivalent attributes are the same between objects. This fixes common mistakes like adding a trailing space by accident, or accidentally writing a full-width character on a Chinese, Japanese, or Korean language keyboard when a half-width character was intended. This option is enabled by default in new projects and it applies Unicode NFC normalization to all model attribute values, replaces full-width characters with half-width characters, and trims the attributes for trailing spaces. For example, this replaces "Ｂ００１" -> B001, "B001 " -> "B001", and "café" -> "café" (these both look the same, but the underlying characters are different)

Note: This step is run after all other transformations are performed. If you wish to customize this behavior, set the option to **No** and use a custom attribute script instead.

- **Automatically publish** – If you want to publish the models automatically after all the model uploads are complete, select **Yes**.
- **Use model defined colors** – Set to **Yes**, if you want to publish the model using the default highlight and selection colors defined by the model. Set to **No**, if you want to define custom highlight and selection colors using the color picker.

  None of the color changes will take effect until the model is published again, regardless of the selected option.

If you have multiple models but want to publish only one, disable the other models. Perform the following for all the models you do not want to publish:

a. Click the name of the model importer.
b. Click **Edit**.
c. Set **State** to **Disabled**.
d. Click **Save**.

6. Click **Publish** to publish the model to users.

   A progress bar displays the publishing progress and an estimate of how much time is left, based on how long the previous publishing took.

   If you click **Cancel**, the uploaded model is discarded and the last uploaded model remains in use.

Results

If you published the uploaded model, you can click **Model** in the main toolbar to open the model in the 3D viewer.

If you did not publish the model, it will remain pending and it will not be visible to users.

7.3.1.4. Republishing a 3D model

If you change the settings of a published model, depending on the type of settings the changes might not be applied until you republish the model. If also a pending (uploaded) model is present at this time, republishing will remove the pending model. When republishing is completed, users might be prompted to refresh the model in their viewer.

Note: Having submodels configured in the project can affect the publishing time as they are published as part of the publishing process.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model Configuration view opens.
3. Change the model settings as appropriate, and then click **Publish**.

   A progress bar displays the publishing progress and an estimate of how much time is left, based on how long the previous publishing took.

7.3.1.5. Exporting and importing model publishing configuration

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. To export settings, click **Export** in Model Publishing pane, and then copy the settings from the **Export Publishing Options** text box.
4. To import settings, click **Import** in Model Publishing pane, paste the required settings into the **Import Publishing Options** text box, and then click **Import**.

Related topics

Submodels

7.3.2. Creating an attribute configuration file

Note: An attribute configuration file is available in importing drawings (DGN, DWF, DWFx, DWG, DXF), IFC models (IFC, IFCzip), or Navisworks models.

An attribute configuration file can be used in importing drawings or models to Cadmatic eShare. The importer will use the configuration file to configure the object attributes described in the configuration file, and discard the attributes which are not configured in the file.

An initial config file can be exported in Cadmatic eBrowser using **Export sample mapping file** option in all compatible eBrowser model importers. The sample mapping file contains all attributes in the imported file and can be edited to create the needed attribute mapping. See eBrowser documentation for more information.

The attribute configuration file syntax:

- Each line of the file defines a single configuration from the first field to the second field.
- Fields are separated by a single unescaped comma.
- Escaping is done by prepending a special character with a backslash.
- Commas, backslashes, and preceding and trailing spaces are to be escaped.
- Lines beginning with '#' are ignored; the original attribute beginning with # must be escaped.

Examples

original , mapped – maps "original" to "mapped"

some attribute, another attribute – maps "some attribute" to "another attribute"

ab\,cd , ef\,gh – maps "ab,cd" to "ef,gh"

Cadmatic

Related topics

Creating a model importer

Model upload and publishing

7.3.3. Attribute settings

In the Attribute Settings view, project administrator can select which object attributes the users see in the 3D model viewer, generate new attributes, and modify all the attributes of an object via scripting. Attributes can also be exported for import to other projects.

See Tutorials for How to configure attribute settings.

7.3.3.1. External attributes

You can edit external attribute definitions as abbreviation-display name pairs by hand for projects without a 3D model or before a model has been published. External attribute definitions take effect at the next publish. If an abbreviation also exists in the model or is produced by a transformation, the model definition takes precedence.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Settings**.
4. Click **External**.
5. In the External Attribute Definitions section, either
    a. click **Add new** to create a new definition and add Abbreviation and Display name to corresponding fields, or
    b. click **Load from Excel or CSV** to add attribute definitions in bulk as an XLSX or CSV file. The file should have two columns: "Abbreviation" and "Display name". Each row represents one attribute definition.

    You can delete individual rows by selecting .
6. Click **Save**.

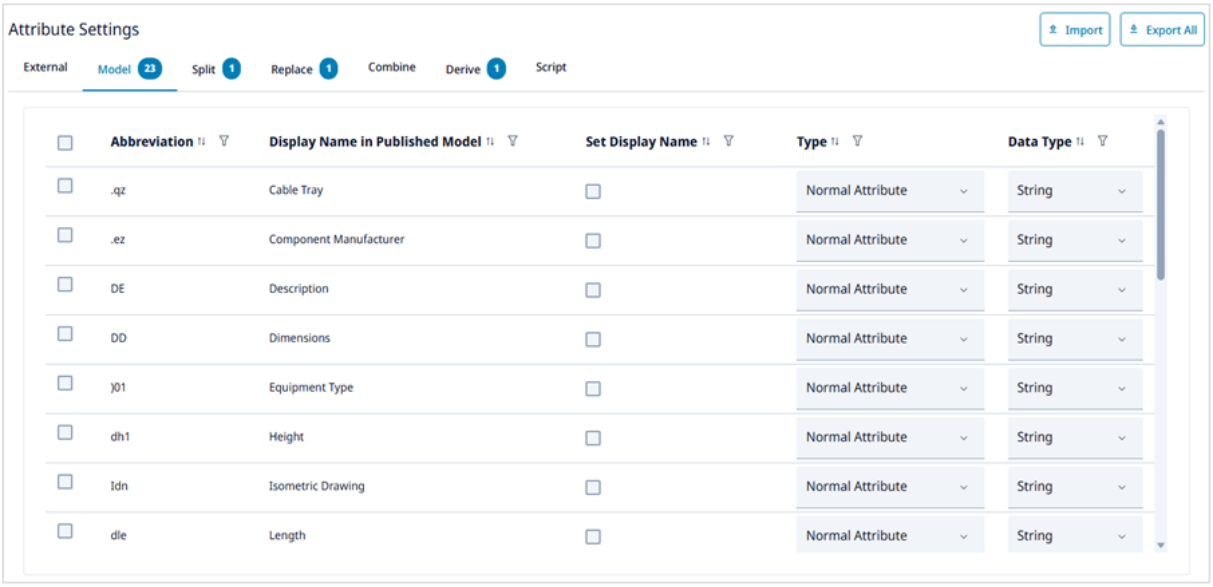
7.3.3.2. Attributes from model file

You can edit settings that affect how users see attributes in the 3D model viewer.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Settings**. The **Model** section lists the attributes that are defined for the 3D model.

   The attributes can be filtered and sorted according to any of the columns. You can select multiple rows with the checkboxes or by holding down Shift key while selecting, and then change the Type and Data Type of all the selected attributes at once.



Note: Attributes related to keeping models separate when publishing (*_origModel*, *Original Model*), bounding box center (*bndX*, *Bounding box center x*; *bndY*, *Bounding box center y*; *bndZ*, *Bounding box center z*), bounding box min (*minX*, *Bounding box minimum x*; *minY*, *Bounding box minimum y*; *minZ*, *Bounding box minimum z*), and bounding box max (*maxX*, *Bounding box maximum x*; *maxY*, *Bounding box maximum y*; *maxZ*, *Bounding box maximum z*) are visible only when they are enabled in Model Publishing configuration. See Uploading and publishing a 3D model.

---

4. Abbreviation and Display Name in Published Model are identifying attributes that must be unique within a project. You can change the display name of an attribute by selecting its check box in the Set Display Name column and then providing the new name in the text field.

5. In the Type column, select how objects are treated when a user searches for objects with specific attributes or attribute values.
    - **Normal Attribute** — All objects are handled separately, and users can search for objects that contain a specific attribute value. For example, if ten objects have the attribute Width = 500, the search *Width equals 500* lists the ten objects, and clicking an individual search result displays the relevant object in the model viewer.
    - **Group Defining Attribute** — All objects that have the same attribute value belong to the same group, and users can search for groups of objects that have the same attribute value. For example, if ten objects have the attribute Width = 500, the search *Width (Group) equals 500* returns one result, and clicking that result displays those ten objects in the model viewer.
    - **Key Attribute** — Attribute becomes the key (ID) attribute of the object if the object does not already have a key in the model file. Users can search for the values of this attribute using the "IDs only" (simple search) and "ID Attribute" (model and point search) options of the search tool.

      Note: Each model object receives a key attribute based on the following priority order:

      1. Key attribute generated by an attribute rule or transformation
      2. Key attribute set in the imported model file
      3. Attribute set as "Key Attribute" in the "Attributes from model file" section

      ---
    - **Do Not Import** — Attribute is not included in the published model.
6. In the Data Type column, select what type of data is stored in the attribute. For example, if attribute values only consists of numbers, setting the type to "Numeric" enables users to search for objects whose attribute value is within a specified range.
    - **String** — Attribute values are handled as text strings.
    - **Date** — Attribute values are handled as Unix dates, in the format YYYY-MM-DD HH-MM-SS, instead of the Unix epoch time format. Fields used for selecting a date display a calendar control.
    - **Numeric** — Attribute values are treated as numeric values.
7. Click **Save** to save your changes.

Results

Publishing the 3D model applies the attribute transformations to objects.

7.3.3.3. Derived attributes

In the Attribute Settings view, the Split, Replace, Combine and Derive sections allows you to define rules that generate new attributes from the values of existing attributes. Generating of new attributes can involve one or several of the following steps:

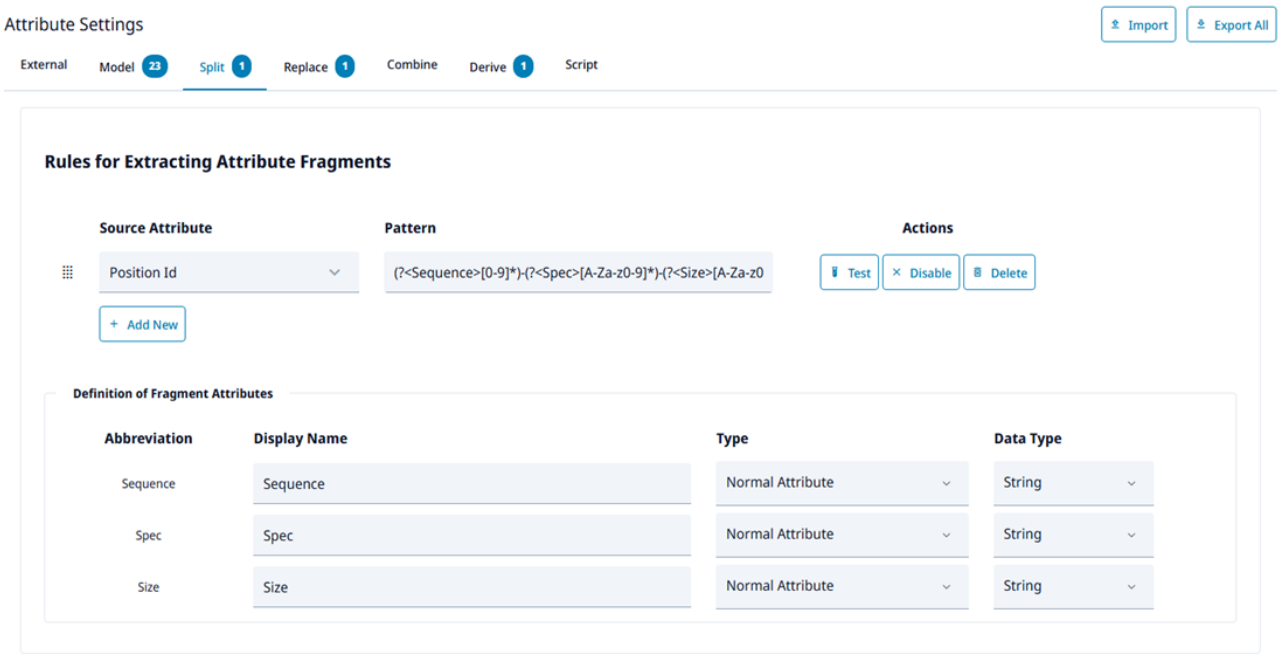
- Extracts parts (text strings) from the value of an existing attribute, as described in Creating attributes by extracting values from existing attributes.
- Replace text strings in the values of existing attributes and in extracted parts, as described in Creating attributes by replacing characters in existing attribute values.
- Combine attribute values, extracted strings, replaced strings, and fixed text strings, as described in Creating attributes by combining existing attributes.
- Create new attributes which are assigned the value of an existing attribute, based on its existence on an object, as described in Creating attributes based on existence of other attributes.

See Regular expressions (regex) – common operators in Tutorials.

7.3.3.3.1. Creating attributes by extracting values from existing attributes

In the Attribute Settings view, the **Split** section allows you create rules for extracting attribute fragments by defining rules that create new attributes by extracting parts from the value of an existing attribute and assigning each extracted value to another attribute. For example, from the attribute value "102-ER-10" you could extract the values "102", "ER", and "10", and assign them to three different attributes.

Extraction is performed using .NET regular expressions that contain "named capture groups". When the rule is applied and the regular expression matches attribute values found in the objects in the 3D model, for these objects a new attribute is created for each named capture group. The name of the capture group is the Abbreviation of the new attribute.



Prerequisites

- You are familiar with creating regular expressions. See About .NET regular expressions.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Settings**. The Attribute Settings view opens.
4. Click **Split**.
5. In the Rules for Extracting Attribute Fragments section, click **Add new**.
6. In the Source Attribute field, select the attribute from which to extract information.
7. In the Pattern field, define the regular expression to use for extracting information.

   Example: To generate a new attribute *cm* whose value is derived from the first three letters of the component manufacturer, considering only manufacturers with alphanumeric characters in the name, select "Component Manufacturer" as the source attribute and specify Pattern as *(?<cm>...)[A-Za-z0-9]*.

Note: Cadmatic AI can be used to assist in creating a regular expression pattern. Select **Cadmatic AI** in the menu to open chat. See Cadmatic AI.

---

8. Click **Test** to test the rule, and review the results. The test is performed on the attributes of a pending model; if a pending model is not present, the test is performed on the attributes of the latest published model.
9. In the Definition of Fragment Attributes section, edit the attribute's Display Name if needed.
10. Select attribute Type to be one of the following:
    - **Normal Attribute** — Attribute values are handled as text strings.
    - **Key Attribute** — Attribute becomes the key (ID) attribute of the object. Users can search for the values of this attribute using the "IDs only" (simple search) and "ID Attribute" (model and point search) options of the search tool.
    - **Group Defining Attribute** — All objects that have the same attribute value belong to the same group. Because the system creates an additional attribute that identifies the group, users can also search for the group itself, in addition to searching for the objects in the group. The Abbreviation of this group attribute is the same as that of the grouped attribute, but in square brackets (*sys* becomes *[sys]*), and its Display Name is the same as that of the grouped attribute, but appended with *(Group)* (*System* becomes *System (Group)*).
    - **Temporary Attribute** — Attribute is used only temporarily, as input for another data transformation; it is not published to the model as a separate attribute.
11. In the Data Type column, select what type of data is stored in the attribute. For example, if attribute values only consists of numbers, setting the type to "Numeric" enables users to search for objects whose attribute value is within a specified range.
    - **String** — Attribute values are handled as text strings.
    - **Date** — Attribute values are handled as Unix dates, in the format YYYY-MM-DD HH-MM-SS, instead of the Unix epoch time format. Fields used for selecting a date display a calendar control.
    - **Numeric** — Attribute values are treated as numeric values.

    You can reorder the rules on the list by dragging and dropping.
12. Click **Save**.

    To disable a rule, select **Disable**. You can enable it again by selecting **Enable**.

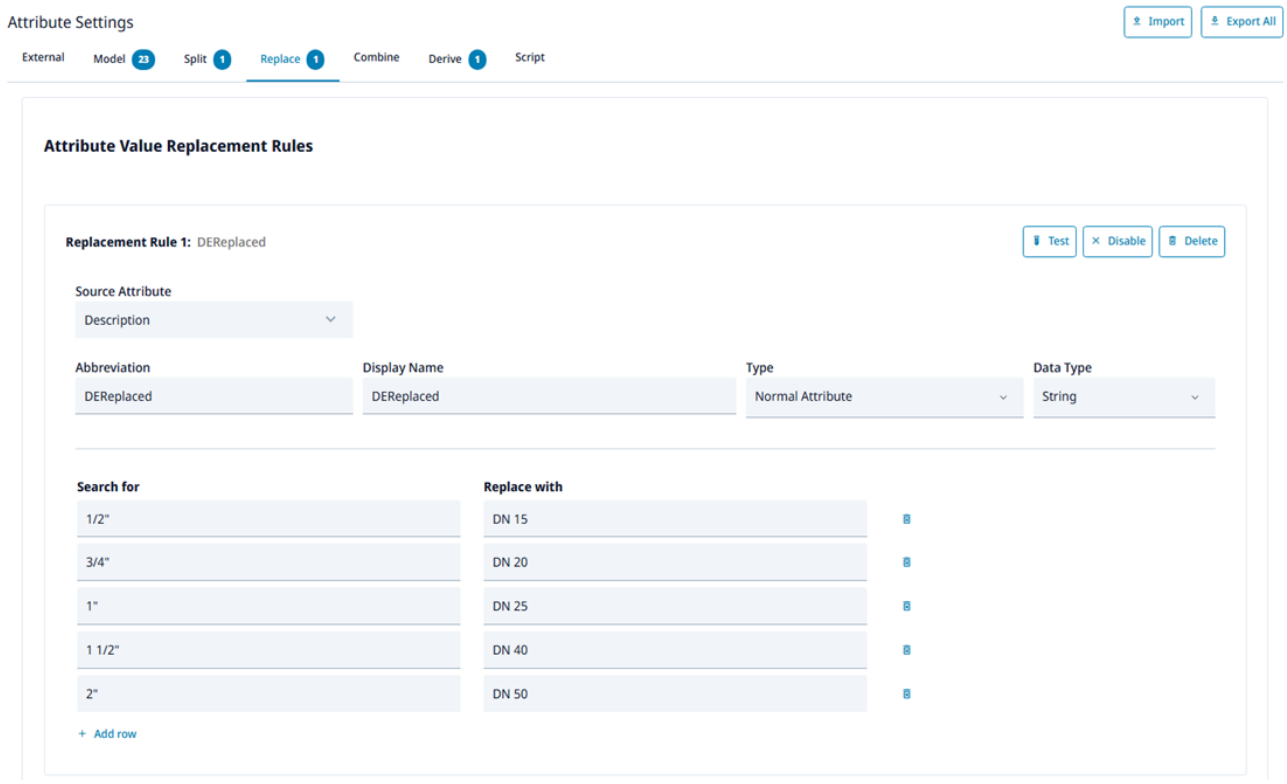
    To delete a rule, select **Delete**.

Results

Publishing the 3D model applies the attribute transformations to objects.

7.3.3.3.2. Creating attributes by replacing characters in existing attribute values

In the Attribute Settings view, the Attribute Value Replacement Rules in **Replace** section allows you to create new attributes by replacing text found in existing attribute values. You can use this for example to remove spaces from the attribute value.



Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Settings**. The Attribute Settings view opens.
4. Click **Replace**.
5. In the Attribute Value Replacement Rules section, click **Add New**.
6. In the Source Attribute field, select the attribute to use in value replacement.
7. Abbreviation and Display Name are by default the same as the abbreviation of the source attribute, but appended with *Replaced* (*sys* becomes *sysReplaced*). Edit if needed.
8. Select the Type of the attribute:

- **Normal Attribute** — Attribute values are handled as text strings.
- **Key Attribute** — Attribute becomes the key (ID) attribute of the object. Users can search for the values of this attribute using the "IDs only" (simple search) and "ID Attribute" (model and point search) options of the search tool.
- **Group Defining Attribute** — All objects that have the same attribute value belong to the same group. Because the system creates an additional attribute that identifies the group, users can also search for the group itself, in addition to the objects in the group. The abbreviation of this group attribute is the same as the abbreviation of the grouped attribute, but in square brackets (*sys* becomes *[sys]*), and its display name is the same as that of the grouped attribute, but appended with *(Group)* (*System* becomes *System (Group)*).
- **Temporary Attribute** — Attribute is used only temporarily, as input for another data transformation; it is not published to the model as a separate attribute.

9. In the Data Type column, select what type of data is stored in the attribute. For example, if attribute values only consists of numbers, setting the type to "Numeric" enables users to search for objects whose attribute value is within a specified range.
   - **String** — Attribute values are handled as text strings.
   - **Date** — Attribute values are handled as Unix dates, in the format YYYY-MM-DD HH-MM-SS, instead of the Unix epoch time format. Fields used for selecting a date display a calendar control.
   - **Numeric** — Attribute values are treated as numeric values.
10. Perform the following as many as times as needed to define the strings to replace:
    a. In the **Search for** field, specify the string to find from existing attribute values.
    b. In the **Replace with** field, specify the replacement string to use in the new attribute.
    c. To add more rows, click **Add row**.

    Examples:
    To replace spaces with underscores, enter a space in the **Search for** field and the underscore character in the Replace field.
    To remove text, enter the text in the **Search for** field and leave the **Replace with** field empty.

Note: The replacements are performed according to the list, from top to bottom. If you have several rules that match the same string, the rule that is highest on the list gets applied first.

---

11. Click **Test** to test the rule, and review the results. The test is performed on the attributes of a pending model; if a pending model is not present, the test is performed on the attributes of the latest published model.
12. Click **Save**.

    To disable a rule, select **Disable**. You can enable it again by selecting **Enable**.

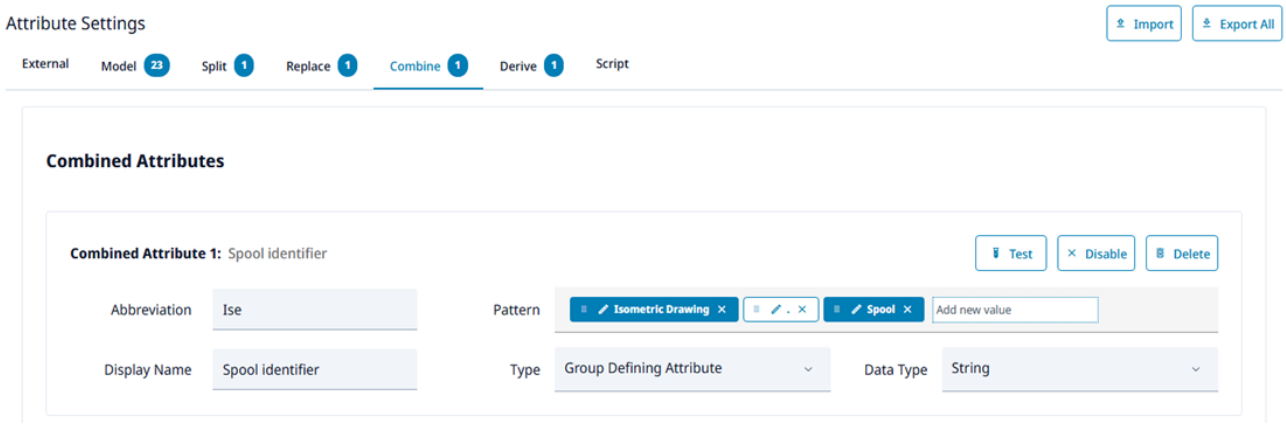
    To delete a rule, select **Delete**.

Results

Publishing the 3D model applies the attribute transformations to objects.

7.3.3.3.3. Creating attributes by combining existing attributes

In the Attribute Settings view, the Combine section allows you to create new attributes by combining the values of multiple attributes and free text.



Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Settings**. The Attribute Settings view opens.
4. Click **Combine**.
5. In the Combined Attributes section, click **Add New**.
6. Enter an Abbreviation and Display Name for the new attribute.
7. Specify the Pattern to use. The pattern can mix attribute names and free text; the values of attributes listed in the pattern will be used to construct the value of the combined attribute.

When you start typing in this field, a list of attributes whose display name matches your input is displayed—select an attribute from the list to add it to the pattern. Typing the exclamation mark (!) lists all available attributes. Press Enter after typing a text string.

Example: To create a new attribute whose value combines the values of the attributes Height and Weight separated by underscores, enter the pattern *Height_Width* by selecting the two attributes from the drop-down menu and typing the underscores in-between and pressing Enter key.



Note: The new attribute will only be added to objects that have all the attributes specified in the pattern.

You can edit or remove an element (attribute or free text) from the pattern using the edit and remove icons.

8. Select the Type of the attribute:
   - **Normal Attribute** — Attribute values are handled as text strings.
   - **Key Attribute** — Attribute becomes the key (ID) attribute of the object. Users can search for the values of this attribute using the "IDs only" (simple search) and "ID Attribute" (model and point search) options of the search tool.
   - **Group Defining Attribute** — All objects that have the same attribute value belong to the same group. Because the system creates an additional attribute that identifies the group, users can also search for the group itself, in addition to the objects in the group. The abbreviation of this group attribute is the same as the abbreviation of the grouped attribute, but in square brackets (*sys* becomes *[sys]*), and its display name is the same as that of the grouped attribute, but appended with *(Group)* (*System* becomes *System (Group)*).
   - **Temporary Attribute** — Attribute is used only temporarily, as input for another data transformation; it is not published to the model as a separate attribute.
9. In the Data Type column, select what type of data is stored in the attribute. For example, if attribute values only consists of numbers, setting the type to "Numeric" enables users to search for objects whose attribute value is within a specified range.
   - **String** — Attribute values are handled as text strings.

- **Date** — Attribute values are handled as Unix dates, in the format YYYY-MM-DD HH-MM-SS, instead of the Unix epoch time format. Fields used for selecting a date display a calendar control.
- **Numeric** — Attribute values are treated as numeric values.

10. Click **Test** to test the rule, and review the results. The test is performed on the attributes of a pending model; if a pending model is not present, the test is performed on the attributes of the latest published model.
11. Click **Save**.

    To disable a rule, select **Disable**. You can enable it again by selecting **Enable**.

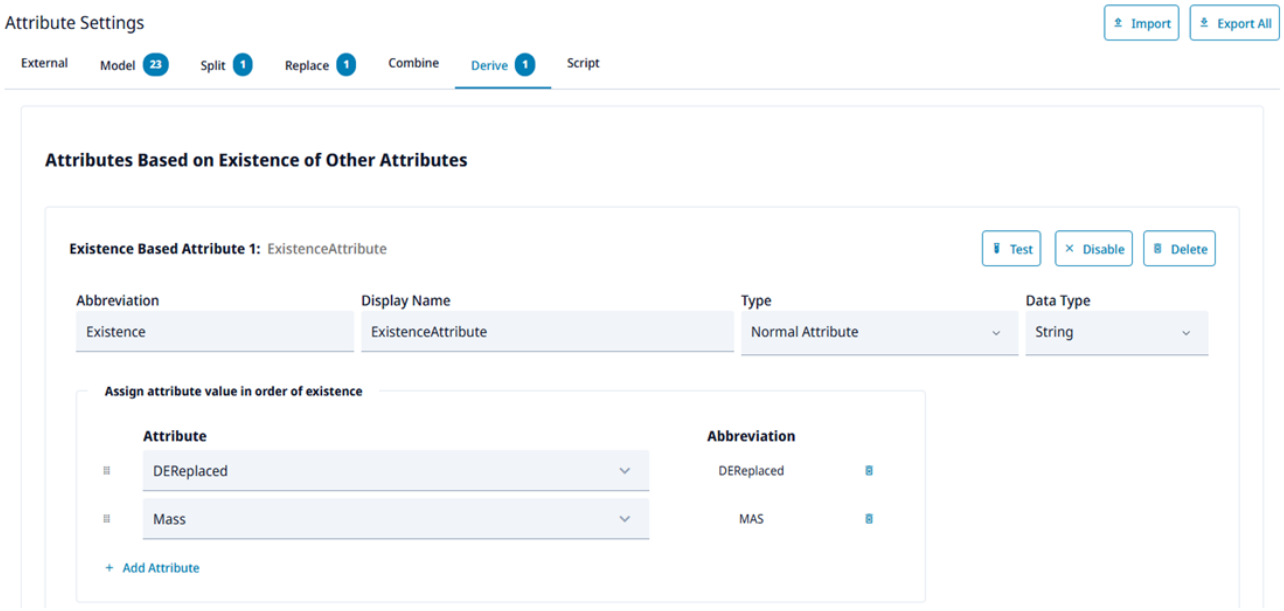
    To delete a rule, select **Delete**.

Results

Publishing the 3D model applies the attribute transformations to objects.

7.3.3.3.4. Creating attributes based on existence of other attributes

In the Attribute Settings view, the Derive section allows you to create new attributes with value assigned by an attribute existing in the model, based on the existence of attributes.



Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.

2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Settings**. The Attribute Settings view opens.
4. Click **Derive**.
5. In the Attributes Based on Existence of Other Attributes section, click **Add New**.
6. Enter an Abbreviation and Display Name for the new attribute.
7. Select the Type of the attribute:
    - **Normal Attribute** — Attribute values are handled as text strings.
    - **Key Attribute** — Attribute becomes the key (ID) attribute of the object. Users can search for the values of this attribute using the "IDs only" (simple search) and "ID Attribute" (model and point search) options of the search tool.
    - **Group Defining Attribute** — All objects that have the same attribute value belong to the same group. Because the system creates an additional attribute that identifies the group, users can also search for the group itself, in addition to the objects in the group. The abbreviation of this group attribute is the same as the abbreviation of the grouped attribute, but in square brackets (*sys* becomes *[sys]*), and its display name is the same as that of the grouped attribute, but appended with *(Group)* (*System* becomes *System (Group)*).
    - **Temporary Attribute** — Attribute is used only temporarily, as input for another data transformation; it is not published to the model as a separate attribute.
8. In the Data Type column, select what type of data is stored in the attribute. For example, if attribute values only consists of numbers, setting the type to "Numeric" enables users to search for objects whose attribute value is within a specified range.
    - **String** — Attribute values are handled as text strings.
    - **Date** — Attribute values are handled as Unix dates, in the format YYYY-MM-DD HH-MM-SS, instead of the Unix epoch time format. Fields used for selecting a date display a calendar control.
    - **Numeric** — Attribute values are treated as numeric values.
9. In the Assign attribute value in order of existence section, click **Add Attribute**.
10. Select which attributes are to be used as a value for the new attribute, if they exist on an object. The rule will check the attributes one by one in the exact order listed in this section. The first attribute on the list will be checked for existence and if it exists on an object, its value is assigned to the new attribute. If the first attribute does not exist, the existence of the second attribute on the list is checked until an attribute is found or the end of list is reached.

You can reorder the attributes on the list by dragging and dropping.

11. Click **Test** to test the rule, and review the results. The test is performed on the attributes of a pending model; if a pending model is not present, the test is performed on the attributes of the latest published model.
12. Click **Save**.

    To disable a rule, select **Disable**. You can enable it again by selecting **Enable**.

    To delete a rule, select **Delete**.

Results

Publishing the 3D model applies the attribute transformations to objects.

7.3.3.4. Script attributes

In the Attribute Settings, the **Script** section allows you to create a C# language script that mass-modifies object attributes when the 3D model is published. The script is applied to original object attributes in the model as well as to derived attributes generated by eShare. You can use the script to automatically perform any of the following tasks:

- Create new attributes
- Change attribute values
- Remove attributes

When writing the script, you can use all the features of the .NET namespaces *System*, *System.Linq*, and *System.Globalization*. The script is executed once per object, and the attributes of the object are available in code in a structure called *attributes*.

The script editor provides basic syntax highlighting and code completion, and it allows you to check whether your script can be compiled. You can get sample results from running the script on the objects of the 3D model, but actual changes to model objects will not be made until the 3D model is published, either manually or by Plant Modeller.

7.3.3.4.1. Using a script for attribute transformations

Perform the following to modify the attributes of objects in the 3D model via a script.

Prerequisites

- You are familiar with the C# programming language and its conventions. See Information for script writers.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Settings**. The Attribute Settings view opens.
4. Click **Script**.
5. In the Script Attributes section, click **Add script**. The script editor is displayed.

   Note: Cadmatic AI can be used to assist in creating a script. Select **Cadmatic AI** in the menu to open chat. See Cadmatic AI.

6. If your script code will introduce completely new attributes, you have to define them first, otherwise they will not show up in the results.

   Click **Add Attribute**, enter the attribute abbreviation and display name, and select whether it is a normal attribute or a group-defining attribute.
7. Write the script code in the script editor.
8. Click **Compile** to check that your script is syntactically correct. Runtime errors might occur even if the script can be compiled. Any errors that occurred are listed on-screen.
9. Click **Test**. This compiles the script and then starts testing the attribute transformations on the objects of the 3D model. All other attribute transformations are applied first, before the script is run on a given object. Objects are processed in this way until one of the following criteria is met:
   - 5 objects have been modified (if possible) and 5 objects remain unchanged by the transformations.
   - Script has encountered 200 errors.
   - There are no more objects in the model.

   The results of the test are shown below the script editor.

   For modified objects the test results table displays four columns:

Abbreviation	Attribute's abbreviation
Display Name	Attribute's display name
Input	Existing attribute value
Output	New attribute value

Each row of the results table represents a single attribute of the object. The table is arranged so that first it lists attributes that have an input value but no output value, and then attributes that have both input and output values. Finally, it lists attributes that only have an output value.

Key attributes are marked with a key icon . If the script modifies an existing attribute value, a conflict icon  is displayed in the Input column.

For unmodified objects (non-matching results) the test results table is otherwise similar, but the Output column and the conflict icon are not shown.

10. Click **Save**.

    To disable an attribute, select **Disable**. You can enable it again by selecting **Enable**.

    To delete an attribute, select **Delete**.

Results

If the saved script can be compiled, whenever the 3D model is published the attribute transformations are applied to the applicable objects.

If the script encounters an error when processing an object, then that object will remain unchanged. The first error is written to the eShare log as an ERROR, and all additional errors are written to the log as TRACE level entries. A large number of errors can cause the publishing to take a long time.

If the script cannot be compiled, it is not applied to the 3D model at all.

7.3.3.4.2. Using a script for setting a GUID Id manually

You can set the GUID Id for an object manually using an attribute script — either directly or based on object attributes. It is also possible to read the original GUID from *attributes.GuidId*.

By default, new GUID Ids are generated in every publish. Using a script to set a GUID Id manually provides a way to generate a unique, persistent GUID Id from the object's attributes, which means it will stay the same from one publish to the next. Administrator needs to configure which attributes are used to generate the GUID Ids.

Example:

Generate a custom GUID Id for the object based on its attributes "vpo", ".n5", and "pli":

```
attributes.SetGeneratedGuidId(attributes["vpo"], attributes[".n5"],
attributes["pli"]);
```

Set the GUID Id manually for the object with vpo 123:

```
if (attributes["vpo"] == "123") {
attributes.SetGuidId("cdb75546-4578-4b35-99af-988dd06d8987");
}
```

If you generate duplicate GUID Ids by accident (eg. two objects with the same attributes), publish will generate warnings in the publish log, and a new GUID Id will be generated for the duplicate object. The publish log persists between server restarts and is not cleared until the next republish.

7.3.3.4.3. Information for script writers

In Cadmatic eShare, you can use the C# programming language to create a script that performs attribute transformations. We recommend that you review the following information before writing attribute transformation scripts.

Language Conventions

C# uses the same conventions as other C-derived programming languages such as C++, Java and JavaScript, as described below.

- *//* marks the beginning of a comment line. Example:
  ```
  // this line does nothing
  ```
- Strings are placed inside quotation marks "". Example:
  ```
  "Equipment", "Air Conditioning"
  ```
- Strings can be combined with the plus character *+*. Example:
  ```
  attributes["type"] = "Lower" + "Right";
  ```
- Statement lines end with the semicolon character *;*. Example:
  ```
  attributes["type"] = "Valve";
  ```
- *==* is equality, *!=* is inequality, *=* is assignment
- Conditions are placed inside parentheses *( )*. Example:
  ```
  if (attributes["sys"] == "Equipment")
  ```
- After a condition, a block of statements is inside curly brackets *{ }*. Example:
  ```
  if(attributes["sys"] == "Equipment")
  { attributes["type"] = "Equipment"; }
  ```
  - You can continue an *if* block with *else*.
  - In conditions, *&&* is *and*, *||* is *or*, *!* is *not*. Example:
    ```
    if(attributes["sys"] == "Equipment" && attributes[".n5"] ==
    "Air Conditioning") { attributes["type"] = "AC"; } else {
    attributes["type"] = "Other"; }
    ```
  - A variable can be defined using the data type: string, int, float, bool. Example:

```
string area;
area = "Store";
int count;
count = 5;
```

- A variable can be defined automatically using *var*, if the data type is defined by the assignment. Example:

```
var count = 5;
var system = attributes["sys"];
```

- An undefined value is called "null".

Attribute existence

You can check whether an object has a given attribute by testing if its value is null. If you try to compare an attribute that does not exist, you will get an error. The following example tests whether the *pli* attribute exists:

```
if(attributes["pli"] != null && attributes["pli"] == "123"){
attributes["chk"] = "Do check";
}
```

Key attribute

The key attribute of an object is defined by *attributes.Key*. It holds the abbreviation of the key attribute of the object. If *attributes.Key* is null, then a key attribute is not defined.

Example:

```
attributes.Key = ".n5";
```

Example:

```
attributes.Key = null;
```

Example:

```
if(attributes.Key == ".n5"){
attributes["type"] = "Equipment";
}
```

Numeric values

All attribute values are strings. If you want to perform mathematical operations on attribute values, you have to convert them to either integers or decimal values. The script is run using the *InvariantCulture* property of .NET by default, which might affect how for example decimal separators are parsed.

Example:

```
if(attributes["fii"] != null){
   float fii;
```

```
    if(float.TryParse(attributes["fii"], out fii)){
        var fii2 = fii * 2;
        attributes["fii"] = fii2.ToString();
    }
}
```

Example:

```
if(attributes["CGX"] != null){
    int xCenter;
    if(int.TryParse(attributes["CGX"], out xCenter)){
      if(xCenter > 0){
          attributes["side"] = "Right";
       } else {
          attributes["side"] = "Left";
     }
   }
}
```

Attribute removal

You can remove an attribute from an object by setting the attribute value to null. Example:

```
attributes["MAS"] = null;
```

Iterating attributes

All the attributes of an object can be iterated. When iterating, the attributes are represented as pairs of an abbreviation (*.Key*) and a value (*.Value*).

You can convert all attributes with a given value, even if the abbreviation is not known.

Example:

```
foreach(var attribute in attributes.Where(a => a.Value ==
"1/2").ToList())
{ attributes[attribute.Key] = "0.5"; }
```

You can remove all attributes with a given value, even if the abbreviation is not known.

Example:

```
foreach(var attribute in attributes.Where(a => a.Value ==
"Undefined").ToList())
{ attributes[attribute.Key] = null; }
```

You can check if an object has a known attribute value, even if you do not know the abbreviation beforehand.

Example:

```
if(attributes.Any(a => a.Value == "Tag1"))
{ attributes["id"] = "Tag 1"; }
```

You can check if an object has an abbreviation with a known beginning.

Example:

```
if(attributes.Any(a => a.Key.StartsWith(".")))
{ attributes["dot"] = "True"; }
```

7.3.3.5. Exporting and importing attribute settings

You can export and import attribute settings in text format. You can use this to copy settings from one project to another.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Settings**. The Attribute Settings view opens.
4. To export settings, click **Export All**, and then copy the settings from the Export Attribute Settings text box.
5. To import settings, click **Import**, paste the required settings into the Import Attribute Settings text box, and then click **Import**.

7.3.3.6. About .NET regular expressions

Regular expressions are a way to define a pattern that is used to search for text strings that match the pattern. They also allow you to extract parts of the text for re-use. There are several varieties of regular expressions; Cadmatic eShare uses the regular expressions of Microsoft .NET, and you can read more about them for example from the following resources:

- http://msdn.microsoft.com/en-us/library/az24scfc(v=vs.110).aspx
- http://www.regular-expressions.info/dotnet.html

See also Regular expressions (regex) – common operators in Tutorials.

Example 1 — Extracting three attributes from a single source attribute value.

```
(?<Sequence>[0-9]*)-(?<Spec>[A-Za-z0-9]*)-(?<Size>[A-Za-z0-9]*)
```

This pattern can be matched to an attribute value that is comprised of three parts separated by dashes (-). The pattern expects the first part to consist of numbers between 0 and 9, and the second and third part to consist of numbers or letters (considering both upper-case and lower-case letters).

For example, if the input attribute value is "102-ER-10", the pattern would extract the following attributes and attribute values:

Attribute	Value
Sequence	102
Spec	ER
Size	10

Note that the pattern requires all three parts to be present. If the input attribute value is "102-ER", the pattern does not generate any new attributes.

Example 2 — Stripping trailing letters from an attribute value.

```
(?<NewLength>.*)[A-Za-z]*$
```

This pattern can be used to drop letters from the end of a source attribute value.

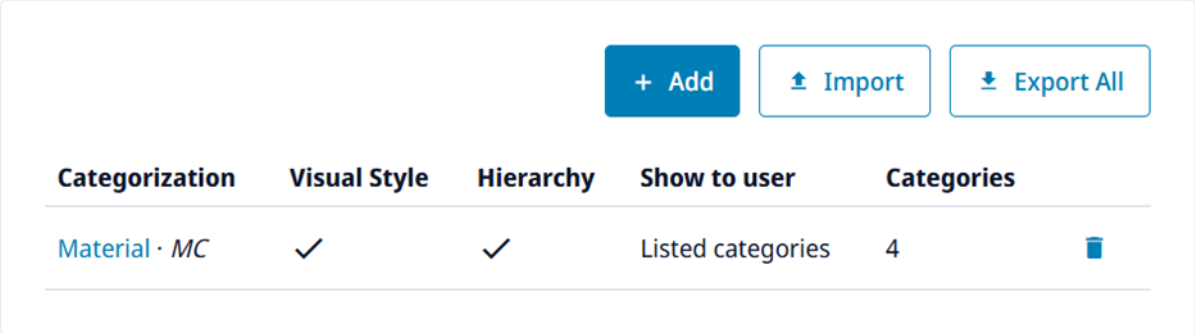
For example, if the input attribute value is "123.45in", the pattern would generate the new attribute NewLength with a value of "123.45".

7.3.4. Attribute categorization

In the Attribute Categorization view, project administrator can define settings that enable the user to easily locate objects that have specific attribute values, for example all objects whose equipment type specifies that they are tank rails. With these settings, objects with specific attribute values can be displayed in the 3D view in a specific color, and the object tree on the Model tab can be arranged according to objects' attribute values. You can define which user groups are allowed to see a given category; system administrators and projects administrators always see all categories.

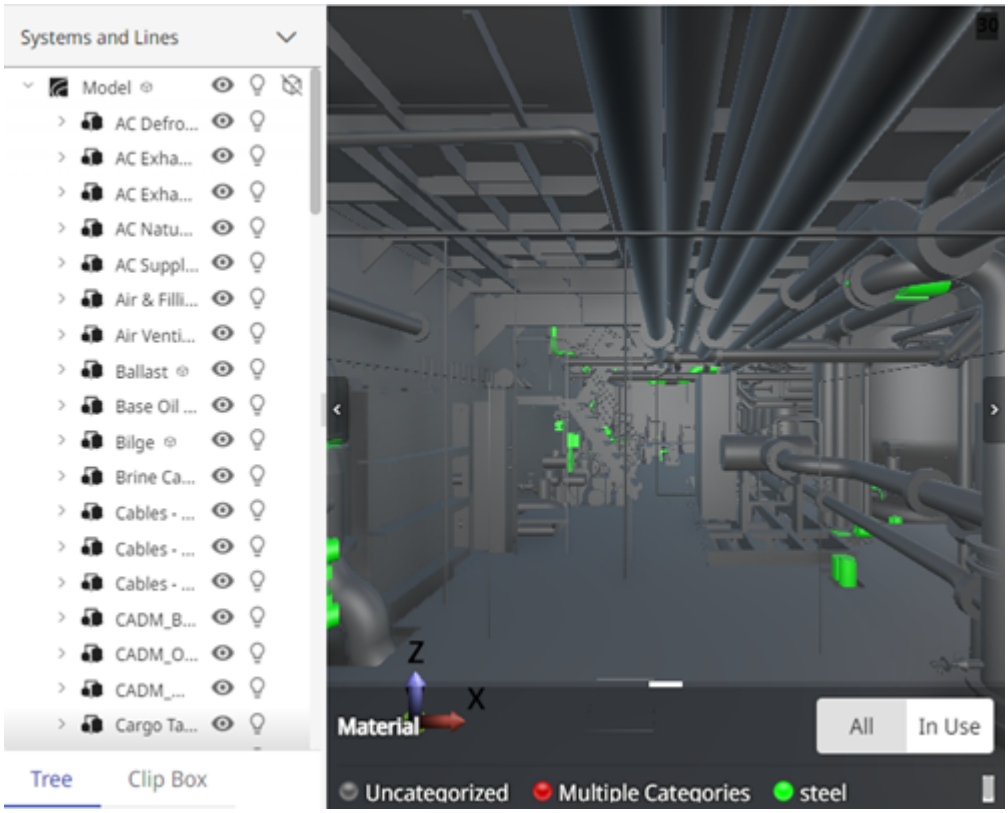
Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model configuration view opens.
3. Click **Attribute Categorization**. The Attribute Categorization view opens, listing the currently defined categories.



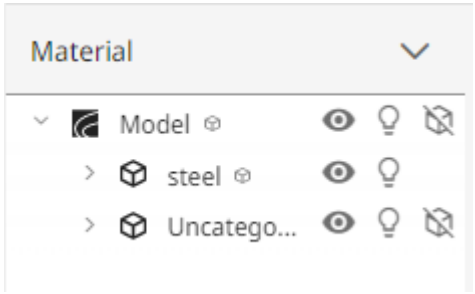
4. Click **Add** and select the required attribute from the list.
5. In the **Name** field, edit the attribute name if needed. This name will be displayed to the user on the Model tab of the 3D viewer, in the visual style and hierarchy drop-down menus. For example, if the attribute name is "Material", you can shorten its display name to "MC".
6. **Visual Style** is enabled by default. When this setting is enabled, the user can select the attribute from the visual style drop-down menu, and the 3D view highlights objects with value-specific colors.

In this example, the user has selected the “MC” category from the visual style menu, and the 3D view highlights the relevant objects with appropriate colors:



7. **Hierarchy** is enabled by default. When this setting is enabled, the user can select the attribute from the hierarchy drop-down menu, and the Model tree lists objects in attribute value specific nodes.

   In this example, the user has selected “Material” from the hierarchy menu, and the Model tree is arranged according to this attribute's values:



8. **Show to user** is set to **Listed categories** by default. This specifies that only the categories that are defined in this view are displayed to the user in the hierarchy and visual style drop-down menus.

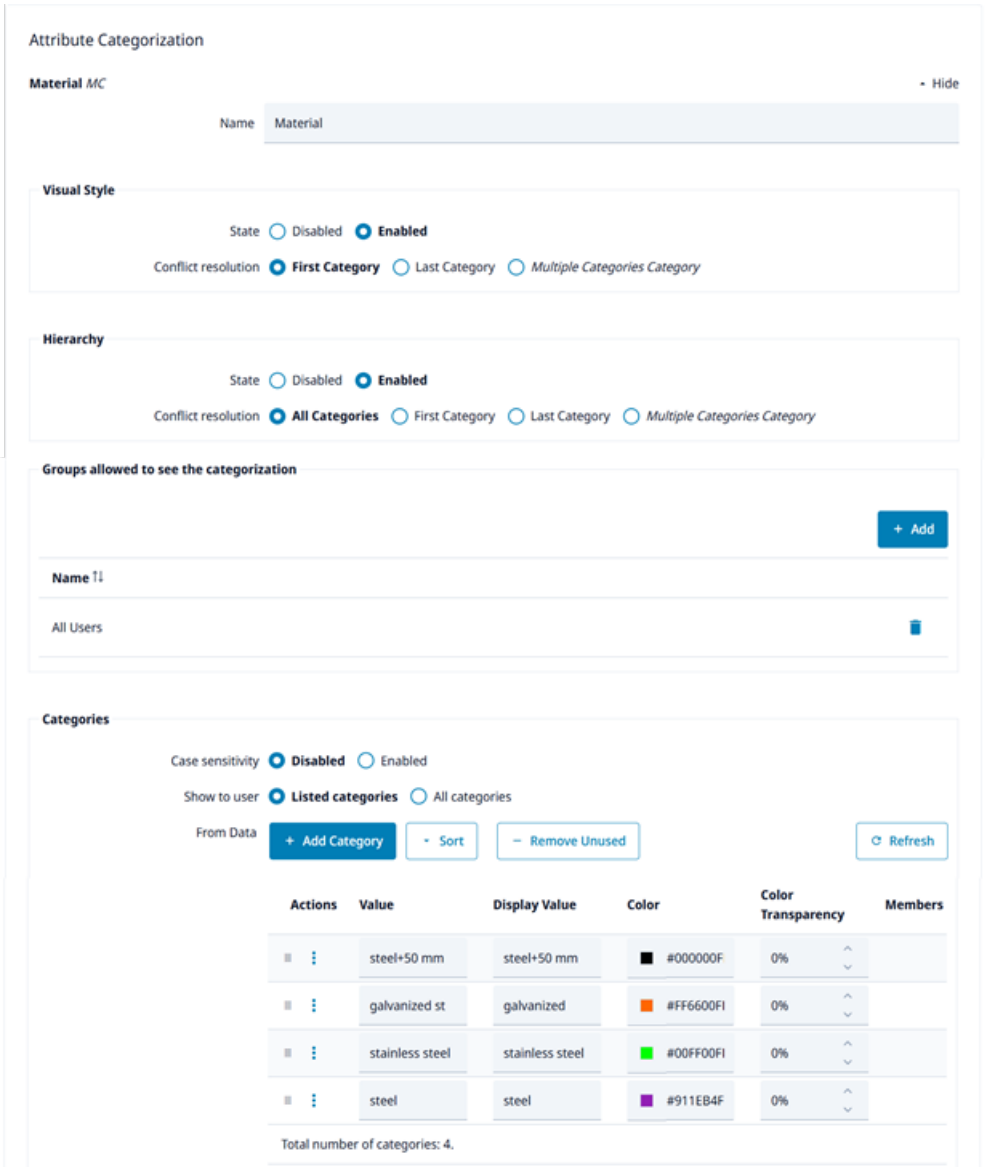
If you set this to **All categories**, the system will create a category for each value that is not defined in this view. Because these dynamic categories are created on-the-fly and not saved, the color that is assigned to them in visual style might change if the values change.

9. In the **Allowed for User Groups** section, select the **Is Allowed** check box of the user groups who should be allowed to see this category in the model viewer. (System administrators and project administrators will see it in any case.)
10. In the **Categories** section, the **Case sensitivity** setting specifies if the value column in the defined categories is expected to be case-sensitive when matching the values to the found values. Case sensitivity does not affect fields that use ranges or regexes (they are always case sensitive). For categorizations created in earlier releases, case sensitivity is disabled by default.

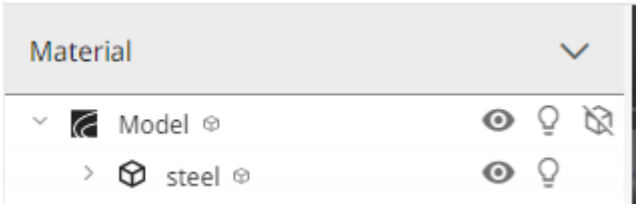
    The **Show to user** setting defines if the user can see only listed categories or all categories.
11. The system displays the number of different values that are assigned to this attribute in the model. Click **Refresh** to see the different values (potential categories), and then specify their display settings as follows:
    - **Value** is the attribute value stored in the model; it can be displayed as a separate category in 3D view if the **Hierarchy** setting is enabled. There should be no need to edit this value; if you do edit it, objects that have the actual attribute value will be listed under “No Category”.
    - **Display Value** will be displayed to the user as the category name, and you can edit it if needed. You can also set the same display value to multiple attribute values, to list them in the same category.
    - If **Visual Style** is enabled, use the **Color** tool to pick an appropriate highlight color for objects that have this attribute value.
    -  – If you do not want a specific attribute value to have neither color-coding nor separate hierarchy, delete its row by clicking the delete button. You are prompted to confirm the action. Objects that have such ignored attribute values will be listed under “No Category”.
    - Change the order of the values in the list by dragging them to the desired order, which is used in the hierarchy view.
12. Click **Save**.

Example

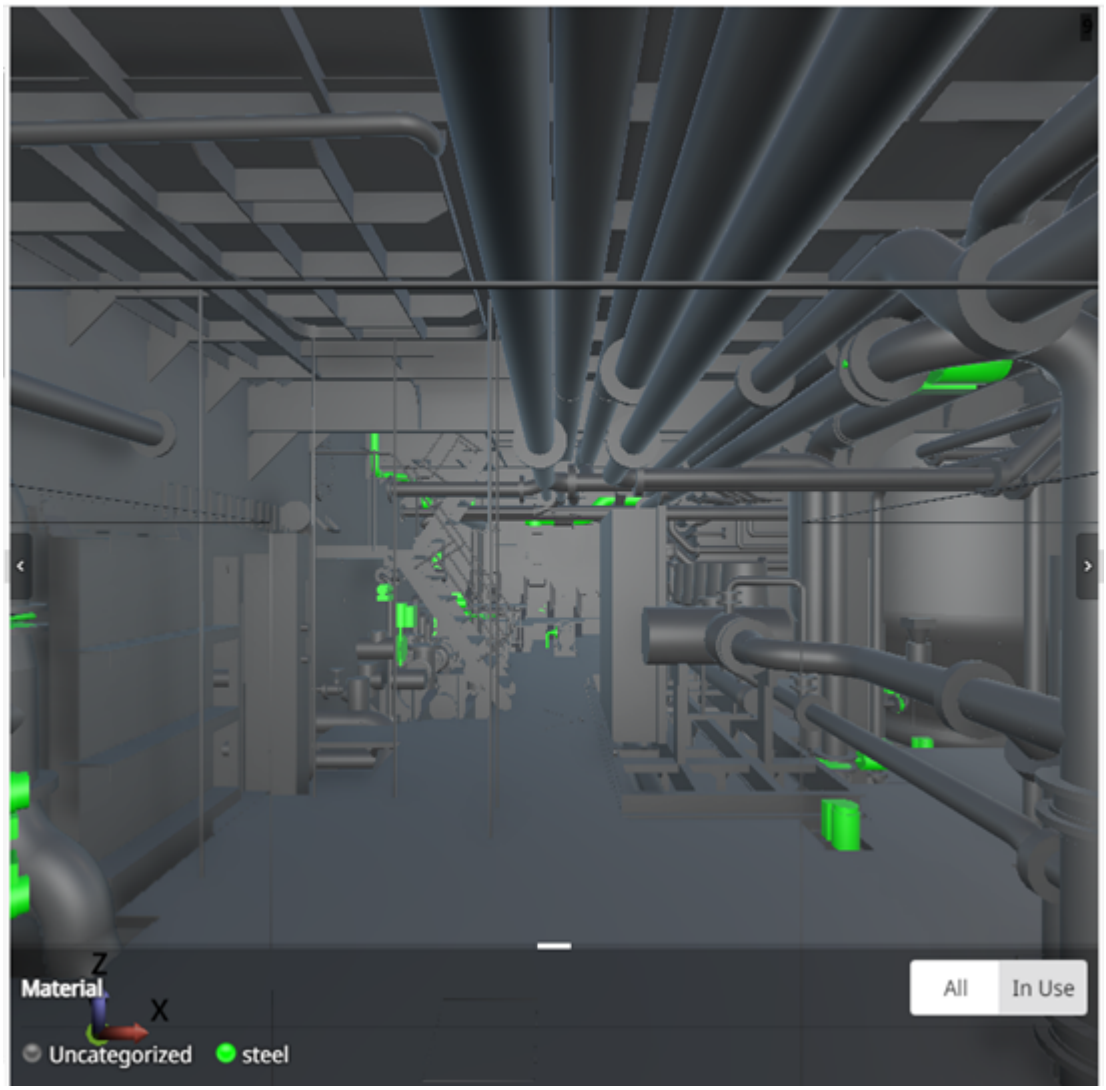
In this example, the Material attribute is given the category name "MC", the Visual Style and Hierarchy settings are enabled for this category, the user group "All Users" is allowed to see this category, and only four attribute values will be included: "copper", "st.", "galvanized steel", and "steel". Because **Show to user** is set to "Listed categories", the system will not generate dynamic categories for any other attribute values.



In the Model tab, if the user selects the Material hierarchy, the objects whose material type is either "st.", "galvanized steel", or "stainless steel" are listed in the "steel" category:

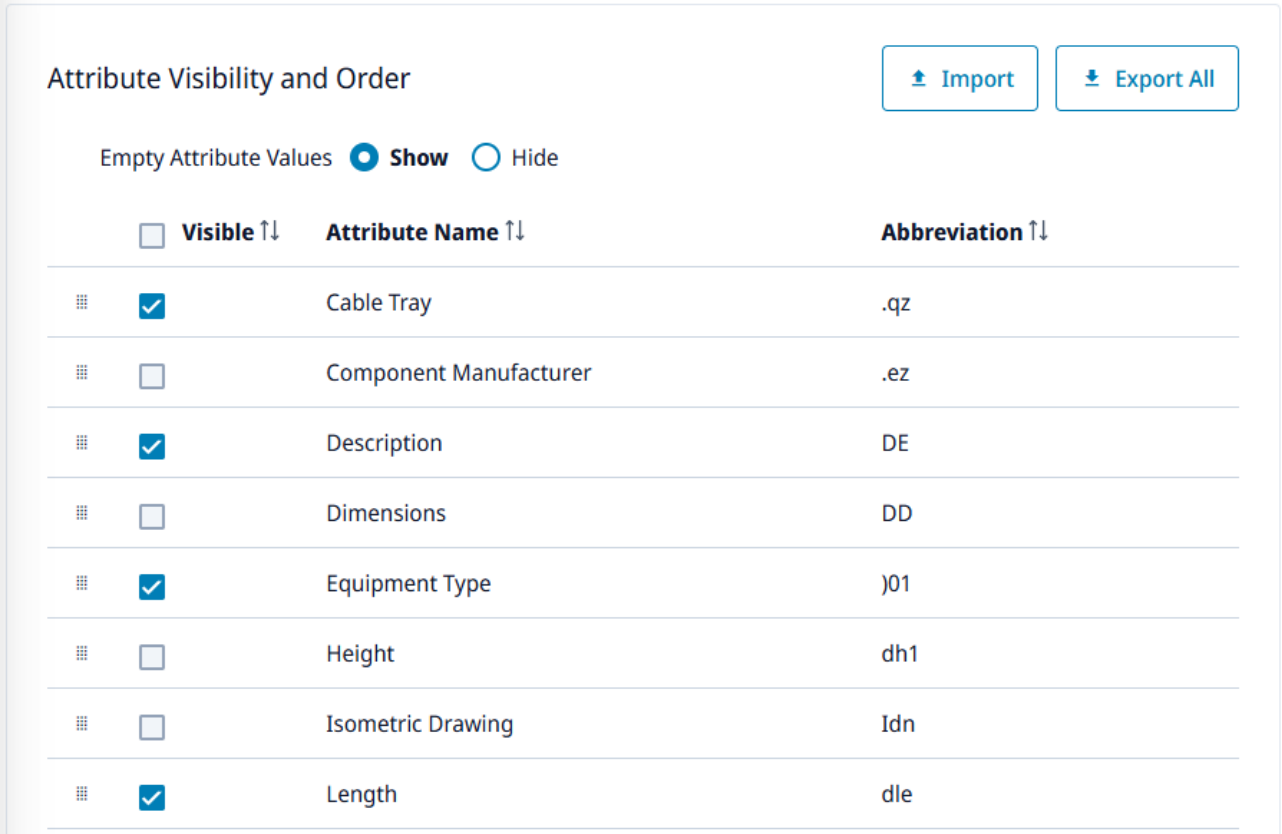


And if the user selects the Material visual style, the objects whose material type is "st.", "galvanized steel", or "stainless steel" are displayed in a single color:



7.3.5. Attribute visibility and order

In the **Attribute Visibility and Order** view, project administrator can define which attributes are visible by default when a user examines an object in the 3D model viewer and the order in which the attributes are listed.



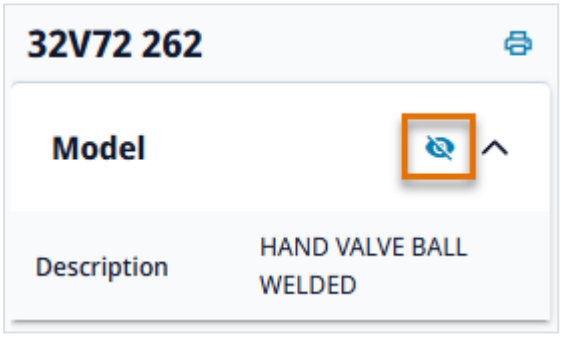
Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Attribute Visibility and Order**. The **Attribute Visibility and Order** view opens.
4. In **Empty Attribute Values** make the following selection:
    - **Show** – This setting enables showing also empty attribute values to the user in eShare
    - **Hide** – This setting hides the empty attribute values from the user in eShare.
5. Select the **Visible** check box of the attributes that should be visible, and clear the ones that should be hidden.

6. You can specify the order in which the attributes are listed. Hold down Shift or Ctrl to select multiple items, or press Ctrl+A to select the whole list, and then either drag or click the appropriate arrow button to move the selected items up or down.
7. To sort the list by visibility, or alphabetically according to attribute name or abbreviation, select the Sort buttons in the columns. Sorting the list by visibility organizes the visible attributes at the top of list.
8. Click **Save**.

Results

When a user selects to examine an object in the model viewer, the properties pane only lists the attributes that are set to be visible. The user can then click the eye button in the **Model** row to also list the hidden attributes.



7.3.6. Model hierarchies

In the Model Hierarchies view, project administrator can add new model hierarchies, and edit, or remove existing model hierarchies.

Model hierarchies are added to the model during publishing. Model hierarchies, which are not new, modified, or disabled are left as they are during publishing.

Original model hierarchies are marked in the **Original** column. The lock symbol indicates that original hierarchies cannot be edited or deleted. A model hierarchy using associations is marked in the **Associations** column.

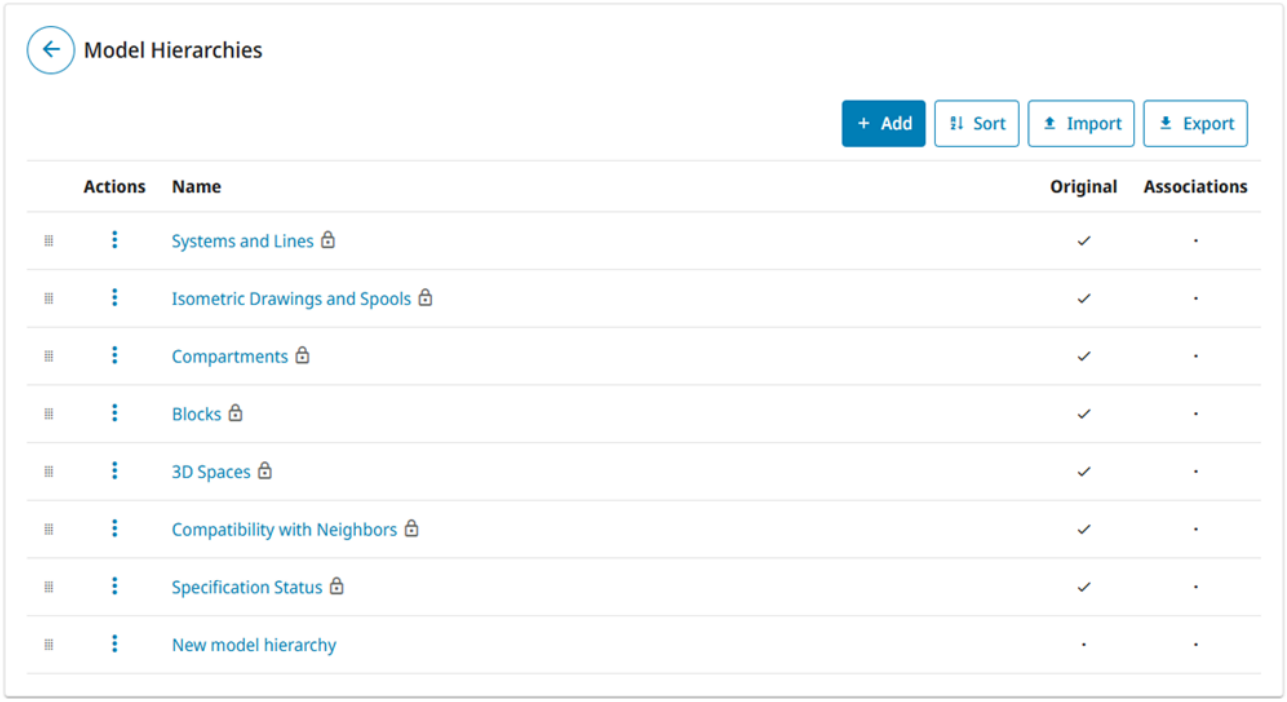
You can sort the model hierarchies in alphabetical or reverse alphabetical order using **Sort**.

Clicking in **Actions** column for a model hierarchy opens a menu with the following options:

- **View** – View information on the model hierarchy.
- **Edit** – Edit the model hierarchy.

- **Clone** – Clone the model hierarchy and open it in edit mode.
- **Delete** – Delete the model hierarchy. A confirmation dialog opens.
- **Deactivate** – Deactivate the model hierarchy.
- **Activate** – Activate the model hierarchy.

Note: Edit and Delete are not available for original model hierarchies.



7.3.6.1. Adding a new model hierarchy

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Model Hierarchies** The Model Hierarchies view opens.
4. Click **Add**.
5. In the **Name** field, enter a descriptive name for the model hierarchy. The name of the model hierarchy must be unique.
6. Specify the following:

- **Active** – Set the model hierarchy as active or inactive.
- **Display unnamed objects** – Specify if unnamed objects are displayed.
- **Use associations** – Specify if the model hierarchy uses associations.
- **Order** – Specify the order for the model hierarchy:
  - **Default** – Use the default order based on hierarchy type. Hierarchies with associations use *Items Order* by default, while other hierarchies use *Alphabetical*.
  - **Original** – Preserve original order.
  - **Items order** – Preserve original order for items and subnodes in Pipelines, Duct lines, and Cable Trays, and for objects on the bottom level of the hierarchy.
  - **Alphabetical** – Sort items on every level alphabetically.

7. In the **Levels** section, click **Add Level**.
8. Select level type:

   If you select **Attributes**, specify the following:
   - **Name** – Enter a descriptive name for the model hierarchy level.
   - **Attributes** – Select attributes, which define the hierarchy level.

     If there are objects, which do not have the defined attributes, these objects are set to the Non-member Category.

   If you select **Categorization**, specify the following:
   - Select category, which defines the hierarchy level.

     If there are objects, which are not included in the defined category, these objects are set to the Non-member Category.

   If you select **Space**, specify the following:
   - Select space, which defines the hierarchy level.

     If there are objects, which are not included in the defined space, these objects are set to the Non-member Category.
9. Click **Add Level** to add more hierarchy levels.

   You can change the order of the hierarchy levels by dragging. To delete a level, click ⋮ in **Actions** for the level and select **Delete**.
10. If the model hierarchy uses associations, in **Included Association Items** section, click **Add Item**.
11. For the association item, specify the following:
    - **Name** – Enter a descriptive name for the association item or top level.
    - **Type** – Specify item type.

- **Subtype** – Specify item subtype.

12. Click **Add Item** to add more association items.
    You can change the order of the association items by dragging. To delete an item, click ⋮ in **Actions** for the item and select **Delete**.
13. Click **Save**.

7.3.6.2. Editing an existing model hierarchy

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Model Hierarchies** The Model Hierarchies view opens.
4. Select ⋮ in Actions and **Edit** in the menu for the hierarchy you want to edit.
5. You can modify names of existing hierarchy levels, or add more attributes. To add more levels, click **Add Level**. To delete a level, click ⋮ in Actions for the level and select **Delete**.
6. Click **Save**.

7.3.6.3. Removing a model hierarchy

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **Model Hierarchies** The Model Hierarchies view opens.
4. Click ⋮ in Actions for the model hierarchy and select **Delete**.
5. Click **Save**.

7.3.6.4. Exporting and importing model hierarchies

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.

3. Click **Model Hierarchies** The Model Hierarchies view opens.
4. To export model hierarchies, click **Export** and copy the data from the text box. Click **Close**.
5. To import model hierarchies, click **Import** and paste the data to the text box. Click **Import**.

7.3.7. Multiselect calculations

In the Multiselect Calculations view, project administrator can define configurations that enable the user to calculate multiple attributes or weight and center attributes in the model view.

You can sort and filter the existing calculations according to name, description, category, and shown when configuration, and depending on the calculated attribute also according to attributes and functions using the buttons in the column header.

Clicking ⋮ in **Actions** column for a multiselect calculation opens a menu with the following options:

- **Deactivate** – Deactivate the configuration.
- **Activate** – Activate the configuration.
- **Edit** – Edit the configuration.
- **Clone** – Clone the configuration and open it in edit mode.
- **Delete** – Delete the configuration. A confirmation dialog opens.

7.3.7.1. Adding calculated attribute configuration

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model configuration view opens.
3. Click **Multiselect Calculations**. The Multiselect Calculations view opens.
4. In **Calculated Attribute** section, click **Add**.
5. In **Add Calculated Attribute** section, specify the following:
    - **Display Name** – Enter a display name for the new attribute.
    - **Active** – Set the calculated attribute active or not.
    - **Show when** – Select if the calculated attribute is shown when at least one selected object has the attribute, or when all the selected objects have the attribute.
    - **Description** – Enter a description for the calculated attribute.

- **Display Category** – Specify the display category. Use the category to group the attributes in the properties pane.
- **Source Attribute** – Select the attribute from which to extract information.
- **Function** – Select which function is applied to the selected attributes: **Average**, **Count**, **Count if missing**, **Count if zero**, **Max**, **Min**,or **Sum**.

  Count if missing can be used to count the number of objects without the selected attribute. This is useful, for example, in verifying that none of the selected objects are missing a weight value.

  Count if zero counts the number of objects with the attribute set to a numeric value of 0.0. It can be used, for example, in verifying that none of the selected objects have been assigned a zero weight value.

6. In **Groups allowed to see the calculated attribute**, select **Add** to add user groups allowed to see the calculated attribute.
   To remove user groups, select .
7. Click **Save** to save the configuration.

7.3.7.2. Adding weight and center attribute configuration

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model configuration view opens.
3. Click **Multiselect Calculations**. The Multiselect Calculations view opens.
4. In **Weight and Center Attribute** section, click **Add**.
5. In **Add Weight and Center Attribute** section, specify the following:
   - **Display Name** – Enter a display name for the new attribute.
   - **Active** – Set the weight and center attribute active or not.
   - **Show when** – Select if the weight and center attribute is shown when at least one selected object has the attribute, or when all the selected objects have the attribute.
   - **Description** – Enter a description for the weight and center attribute.
   - **Display Category** – Specify the display category. Use the category to group the attributes in the properties pane.
6. Select **Weight Attribute** and enter **Weight Attribute Name**.
7. Select **Center X Attribute** and enter **Center X Attribute Name**.

8. Select **Center Y Attribute** and enter **Center Y Attribute Name**.
9. Select **Center Z Attribute** and enter **Center Z Attribute Name**.
10. In **Groups allowed to see the weight and center attribute**, select **Add** to add user groups allowed to see the weight and center attribute.
    To remove user groups, select .
11. Click **Save** to save the configuration.

7.3.7.3. Cloning an attribute configuration

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model configuration view opens.
3. Click **Multiselect Calculations**. The Multiselect Calculations view opens.
4. Select  in Actions and **Clone** in the menu for the configuration you want to clone.
5. Make modifications to the cloned configuration and click **Save**.

7.3.7.4. Editing an existing attribute configuration

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model configuration view opens.
3. Click **Multiselect Calculations**. The Multiselect Calculations view opens.
4. Select  in Actions and **Edit** in the menu for the configuration you want to edit.
5. Make modifications and click **Save**.

7.3.7.5. Removing an attribute configuration

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model configuration view opens.

3. Click **Multiselect Calculations**. The Multiselect Calculations view opens.
4. Select ⋮ in Actions and **Delete** in the menu for the configuration you want to remove.

7.3.7.6. Exporting and importing attribute configurations

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The Model configuration view opens.
3. Click **Multiselect Calculations**. The Multiselect Calculations view opens.
4. To export attribute configurations, click **Export** and copy the data from the text box. Click **Close**.
5. To import attribute configurations, click **Import** and paste the data to the text box. Click **Import**.

7.3.8. eGo QR configuration

In Cadmatic eShare, the user can print the properties of an object or object group. The QR code that is included in these print-outs functions as a link to the entity in the 3D model, and Cadmatic eGo can read the QR code to open the entity in its 3D view.

If your site uses QR codes that are not generated by eShare, you can still enable eGo to read the QR codes—by using a script.

In the eGo QR Configuration view, project administrator can enter a custom script that eGo can use if it fails to read a QR code in the normal way.



The script must be written in C#, and it needs to parse a string-typed variable named *qrCode* (representing the data received from reading a QR code) into a local object named *result*. This local object can have the following properties:

result.Key	The key of the object to be found from the model.	string
result.PoiName	The name of the Smart Point to be found from the model.	string
result.PoiExternalId	The External Id of the Smart Point to be found from the model.	string
result.Abbreviation	The abbreviation of the attribute to be found from the object. (If set, also *result.Value* must be set.)	string
result.Value	The value of the attribute to be found from the object. (If set, also *result.Abbreviation* must be set.)	string
result.CameraX *result.CameraY* *result.CameraZ* *result.CameraSlope*	Coordinate values for the camera position in the 3D view. Also slope and rotation can be set for the view angle. All camera position values are optional.	float

result.CameraRotation		

The script supports regular expressions and Uri-objects, as well as *System.Text* Namespace functions.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Model Configuration**. The model configuration view opens.
3. Click **eGo QR Configuration**. The eGo QR Configuration view opens.
4. Create the script in the editor box.

   Note: Cadmatic AI can be used to assist in creating a script. Select **Cadmatic AI** in the menu to open chat. See Cadmatic AI.

5. Enter some valid test data in the **Test Input** field and click **Test** to evaluate the script. eShare runs the script and shows the output values it received, as well as compilation errors, if there were any.
6. Click **Save** to save the completed script.

Related Tasks

- Use the **Export All** and **Import** buttons of the script configuration view to copy the script code to the projects that require custom QR parsing.
- In Cadmatic eGo, the user must synchronize the required projects with eShare. This downloads the custom QR parsing method from eShare, and then the user can start reading custom QR codes.

7.4. Submodels

A submodel is a part of the 3D model that users can open or download instead of the full model. Submodels are useful when the full model is too large to download or open efficiently, for example on a mobile device or over a slow connection. They are also useful when users only need a specific part of the model, such as one area or deck.

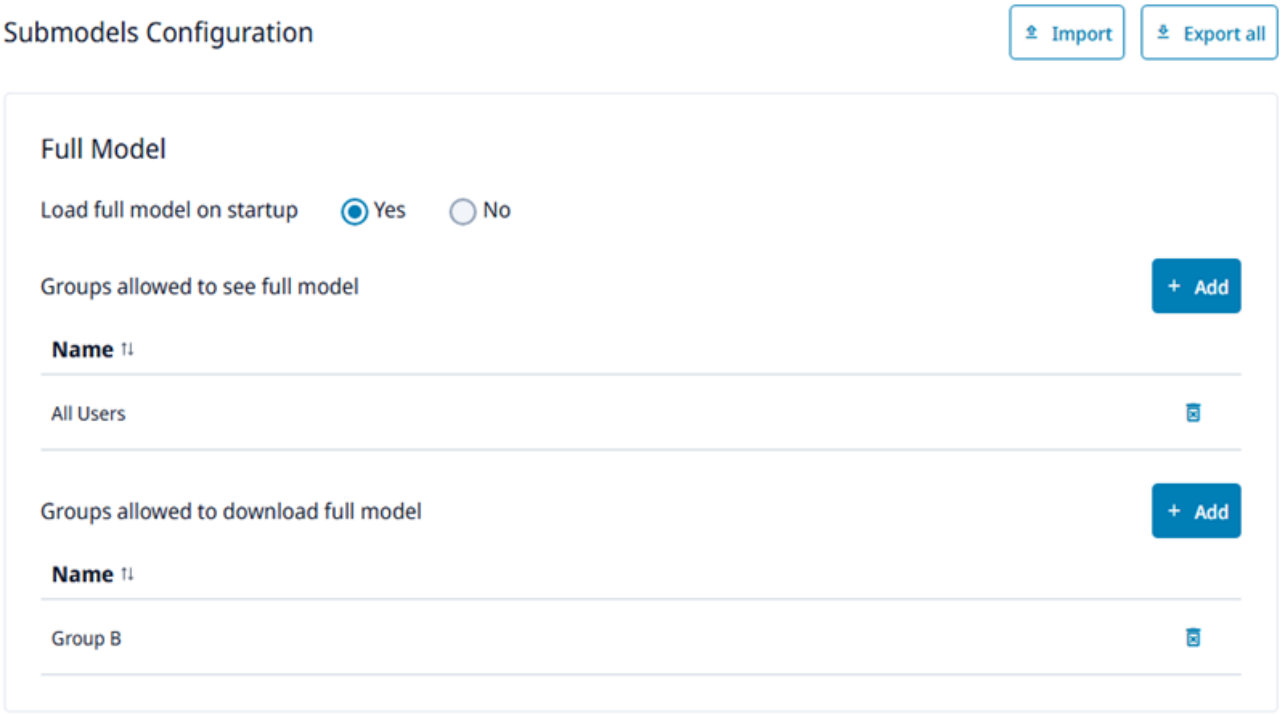
In the Submodel Division Rules view, a project administrator defines how the model is divided into submodels when it is published. Users can then select which submodel to open in eShare or download in eGo.

Project administrators can also control who sees what. They can define which user groups are allowed to view and download the full model, and set permissions for individual submodels. For example, a subcontractor can be given access only to the submodel for their own area.

If you have projects created in 2023T2 containing submodels, they will need to be re-published.

Note: Submodels in the project affect publishing time and a large number of submodels takes more time to publish.

7.4.1. Configuring full model permissions



You can use the **Import** and **Export All**buttons to copy division rules from one project to another.

In **Full model** section, if **Load entire model on startup** is set to **Yes**, the entire model is loaded on startup (selected by default) or if set to **No**, eShare will not automatically load the full model, and will instead show the list of submodels available that can be selected and loaded by the user.

In **Groups allowed to see full model**, you can define which user groups are allowed too see the full model.

If downloading the model is enabled in *Cadmatic.EnableDownloadButton* in Internet Information Settings (IIS) Manager by setting it All or True, **Groups allowed to download full model** is visible and

you can define which user groups are allowed to download the full model, i.e. have **Download Model** button available in Home view.

Download Model button is available only to users who belong to both group(s) with permission to see the full model and to download the full model.

Permissions affect only the geometry; attributes, markups and smart points of hidden objects stay visible.

- If no groups are selected, only administrators can see or download the full model.
- Add user groups with **Add**.
- Remove user groups with delete button.
- If **All Users** group is selected, other group selections will be ignored.
- Save changes with **Save**.

Note: Full model permissions take effect after saving the configuration. The model does not need to be re-published.

7.4.2. Configuring submodel division rules



In **Submodel division rules** section, use the **Add** button to create a new submodel division rule.

Use the edit button to edit a rule and the delete button to remove a rule.

You can use **Edit all** to edit all division rules in one view.

Restricted access column shows if the visibility of the submodels has been restricted to certain user groups.

Permissions affect only the geometry; attributes, markups and smart points of hidden objects stay visible.

Click the down arrow on the row to view a list of submodels found by eShare, their object count, and which groups are allowed to see them, if access is restricted.

The submodels can be created using attribute values, filtering rules, coordinates or a combination of two. Submodel permissions are configured in the submodel division rules.

Note: Submodels will be visible only after the model has been published.

7.4.2.1. Submodel creation by attribute

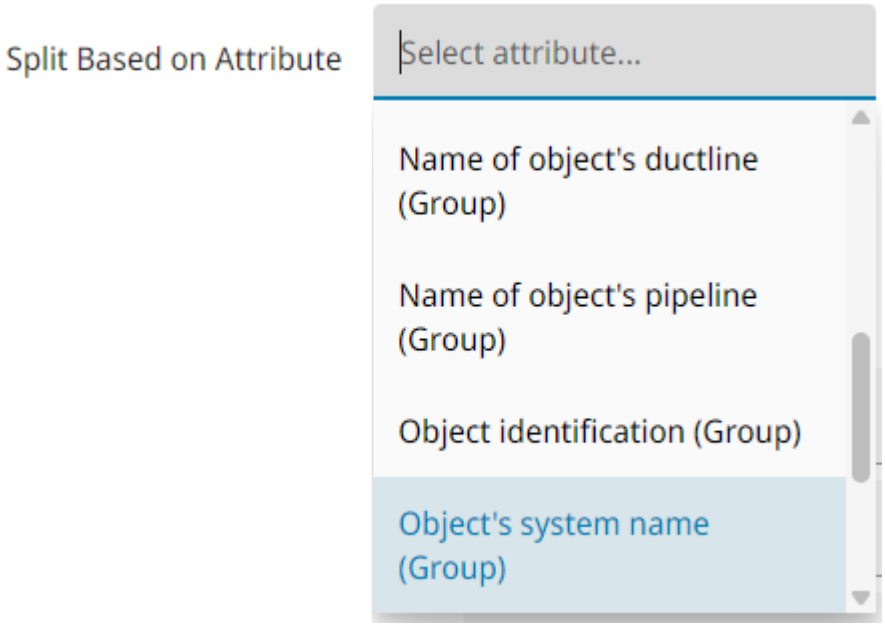
You can split the full model into smaller submodels based on the values of a group-defining attribute. You can use Split Based on Attribute together with Filtering Rules, in which case the rules will act as another filter when generating the submodel(s). If using only split based on attribute, each group of objects that have a specific attribute value forms one submodel.

Prerequisites

- In the Attribute Settings view, the attribute to use is defined as a "Group Defining Attribute". See Attributes from model file.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Submodel**. The Submodel Division Rules view opens.
3. Click **Add**.
4. In the Unnamed Submodel Rule pane, specify the following settings:
    - **Name** – Enter a descriptive name for the rule.
    - **Description** – Enter a description of what the rule will include in the submodel.
5. In the **Split Based on Attribute** field, select the attribute to use.



6. To configure only selected user groups to have access to the submodels, in the permission rules section, select **Add rule**.
7. Enter a regular expression pattern to **Pattern to match** field. The pattern is a regex which matches submodel names, and grants access only to the selected user groups. Users not in any of these user groups will not see the submodels. If there are not any submodels matching the pattern with limited permission, the submodels will be visible to all users. If there are multiple permission rules matching a submodel, a user in at least of one the user groups with access will see the submodel.
8. Select the user groups in **Groups allowed to see** dropdown menu.
9. Click **Save**.

Note: Submodel permissions take effect after saving the configuration. The submodels do not need to be regenerated.

7.4.2.2. Submodel creation by filtering rules

You can filter the submodel or submodels based on filtering rules.

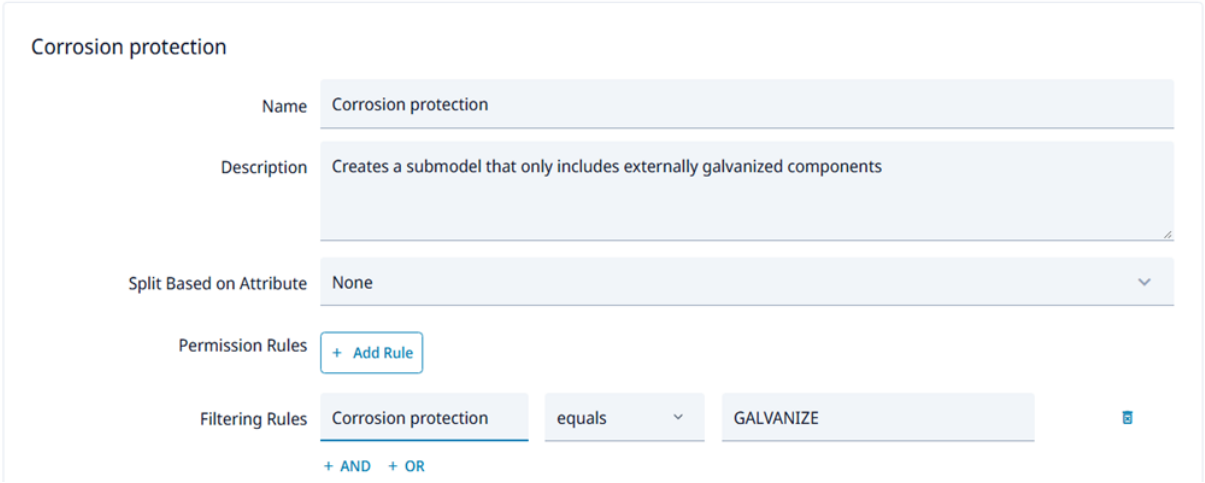
Note: In large models creating an "equals" rule starting with wildcard * may cause the generation to take a long time.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Submodel**. The Submodel Division Rules view opens.
3. Click **Add**.
4. In the Unnamed Submodel Rule pane, specify the following settings.
   - **Name** – Enter a descriptive name for the rule.
   - **Description** – Enter a description of what the rule will include in the submodel.
5. To configure only selected user groups to have access to the submodels, in the permission rules section, select **Add rule**.
6. Enter a regular expression pattern to **Pattern to match** field. The pattern is a regex which matches submodel names, and grants access only to the selected user groups. Users not in any of these user groups will not see the submodels. If there are not any submodels matching the pattern with limited permission, the submodels will be visible to all users. If there are multiple permission rules matching a submodel, a user in at least of one the user groups with access will see the submodel.
7. Select the user groups in **Groups allowed to see** dropdown menu.
8. In the **Filtering Rules** field, select the attribute to use. Additional fields are displayed for defining the criteria, such as whether the attribute exists or has a specific value.
9. Add as many attributes to the rule as needed. The submodel will include only those model objects and groups that match every criteria of the rule.

   You can add terms using **+ AND** or **+ OR**. Using AND adds a new term below the previous term, and creates a query where the previous and the new term(s) must match. Using OR creates a new group of terms, which is a OR query with the previous group of terms. The terms inside the term groups create an AND query.

   In this example, the rule selects an object if the object has the attributes *Corrosion Protection* and the value of *Corrosion protection* is "Galvanized only externally".



10. Click **Save**.

Note: Submodel permissions take effect after saving the configuration. The submodels do not need to be regenerated.

7.4.2.3. Submodel creation by coordinates

You can filter the submodel or submodels based on geometric area by using coordinates. If set, the rule will filter out all objects from the submodel(s) generated by the rule that are not within the set coordinate values.

You can use Split Based on Attribute or Filtering Rules together with Split Based on Coordinates, in which case the coordinates will act as another filter when generating the submodel(s). If used on its own, Split Based on Coordinates will generate a single submodel, which contains all objects within the given coordinates.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Submodels**. The Submodel Division Rules view opens.
3. Click **Add**.
4. In the Unnamed Submodel Rule pane, specify the following settings.
   - **Name** – Enter a descriptive name for the rule.
   - **Description** – Enter a description of what the rule will include in the submodel.
5. To configure only selected user groups to have access to the submodels, in the permission rules section, select **Add rule**.

6. Enter a regular expression pattern to **Pattern to match** field. The pattern is a regex which matches submodel names, and grants access only to the selected user groups. Users not in any of these user groups will not see the submodels. If there are not any submodels matching the pattern with limited permission, the submodels will be visible to all users. If there are multiple permission rules matching a submodel, a user in at least of one the user groups with access will see the submodel.
7. Select the user groups in **Groups allowed to see** dropdown menu.
8. In **Split Based On Coordinates** field, enter the From and To values for X, Y and Z planes.
9. Click **Save**.

Note: Submodel permissions take effect after saving the configuration. The submodels do not need to be regenerated.

7.5. Maps

A model's map represents the layout of the 3D model. eShare displays the map in the 3D model viewer. If the map is not part of the 3D model, a system administrator can import one or more maps to the model. The map can be a DWG, DXF, DGN, JPG, JPEG, PNG, TIF, TIFF, GIF, BMP, or PDF file.

After you have imported the map, define the map settings. You can define if the map should be used for only a specified range of points in the model or for the whole model, and if needed, change the map's name. The map's name can be up to 40 characters long.

If you set a specific range, the map will only show point clouds, Smart Points and Markups that are within that range.

After you have uploaded the maps, you can rearrange them by dragging in the administrator's Map view.

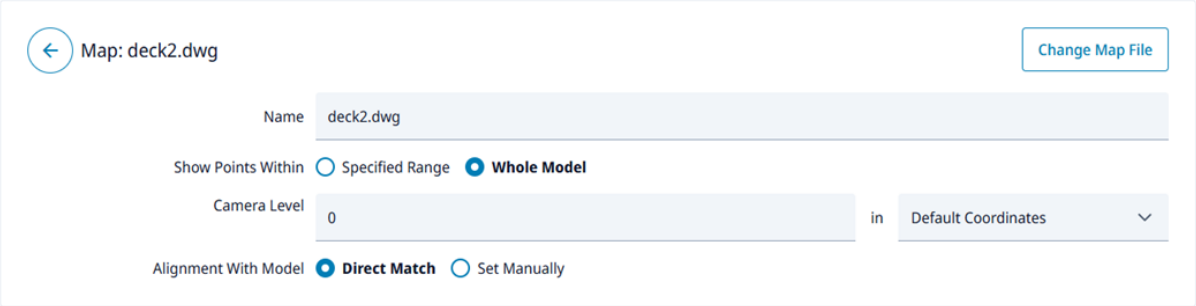
Prerequisites

Make sure that the map file uses the same coordinates as the 3D model. If the coordinates are different, the map and the model are not synchronized.

Plan the range of points that the map should use. You can check the coordinates from the **Model** view. To view the coordinates, enable coordinate display in **Settings > Visualization**. See Visualization.

Do the following:

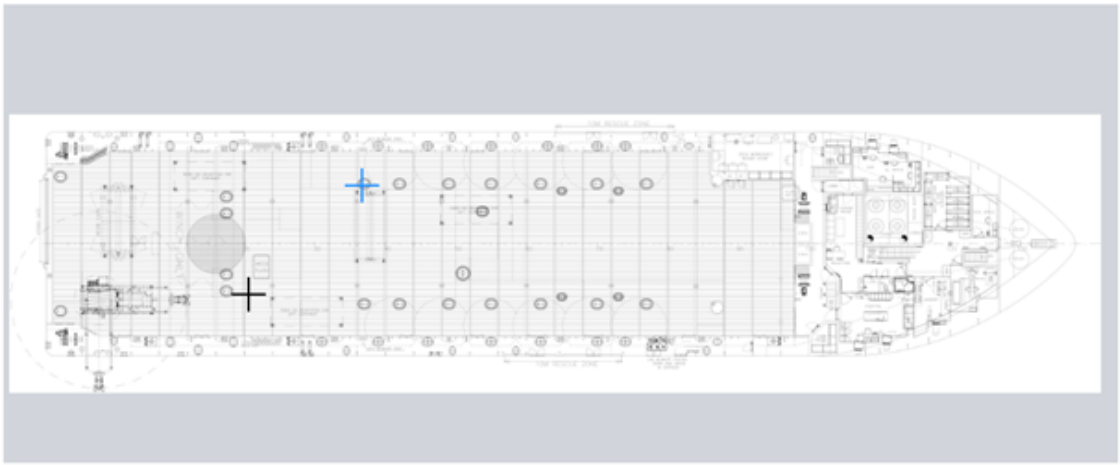
1. Navigate to the project to edit, and then click **Project Admin** in the main toolbar. The project administration view opens.
2. Click **Maps**. The **Map Configuration** view opens.
3. In the **Map Configuration** view, click **Upload Map**.
4. Select the map file from your computer's file system and click **Open**. You can select multiple map files at once with Ctrl or Shift key. eShare loads the map.
5. Edit the map properties. In the **Map Configuration** view, click on the name of the map, or ⋮ button in Actions column and select **Edit**. The **Edit Map** view opens.

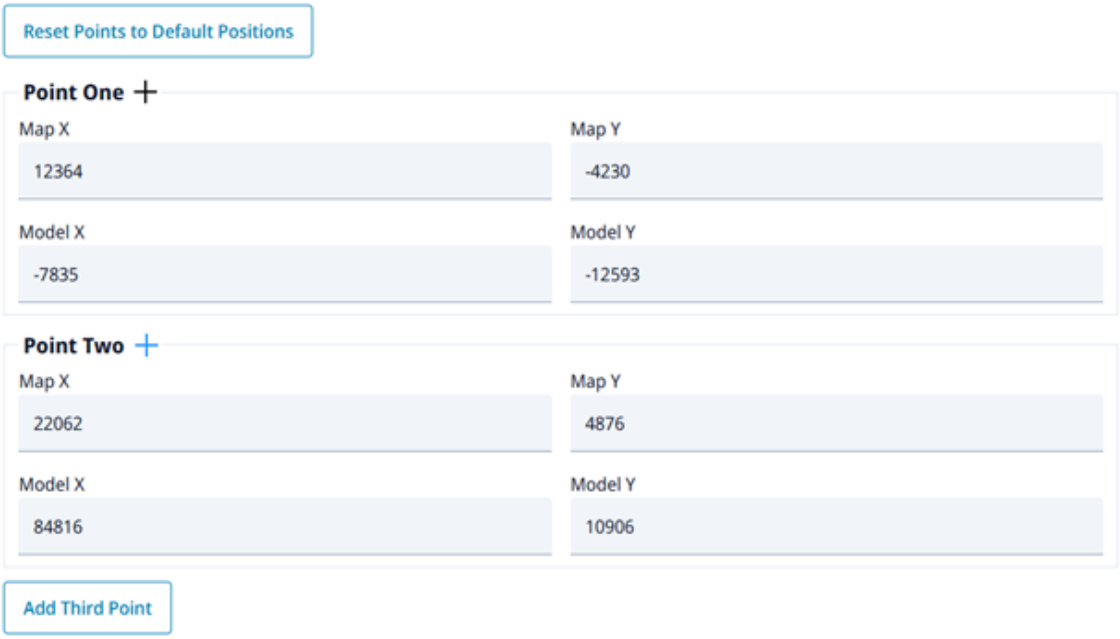


6. Change the name of the map, if needed. By default, eShare uses the file name of the map.
7. Define if the map should show points from a specified range or for the whole model.
   - If the map should show points only for a specified range, define the start Z value in the **Z-Value From** and the end Z value in the **Z-Value To** fields. If the model uses named coordinates, eShare automatically displays them in a drop-down menu. Select the coordinates that you need, or enter the coordinates as a number. You can select the used coordinate system from the drop-down menu.
   - If the map should show points for the whole model, define the camera level in the **Camera Level** field. The camera level is the Z level where the user is taken in the model when they double click on the map.

     If the model uses named coordinates, eShare automatically displays them in a drop-down menu. Select the coordinate that you want to use, or enter the coordinate as a number. You can select the used coordinate system from the drop-down menu.
8. Define the alignment with the model.
   - Select **Direct Match** if the map coordinates match the model directly.
   - Select **Set Manually** if the coordinates do not match and they need to be manually aligned. If the map file is an image (JPG, JPEG, PNG, TIF, TIFF, GIF or BMP file) or a PDF

file, only **Set Manually** can be selected.



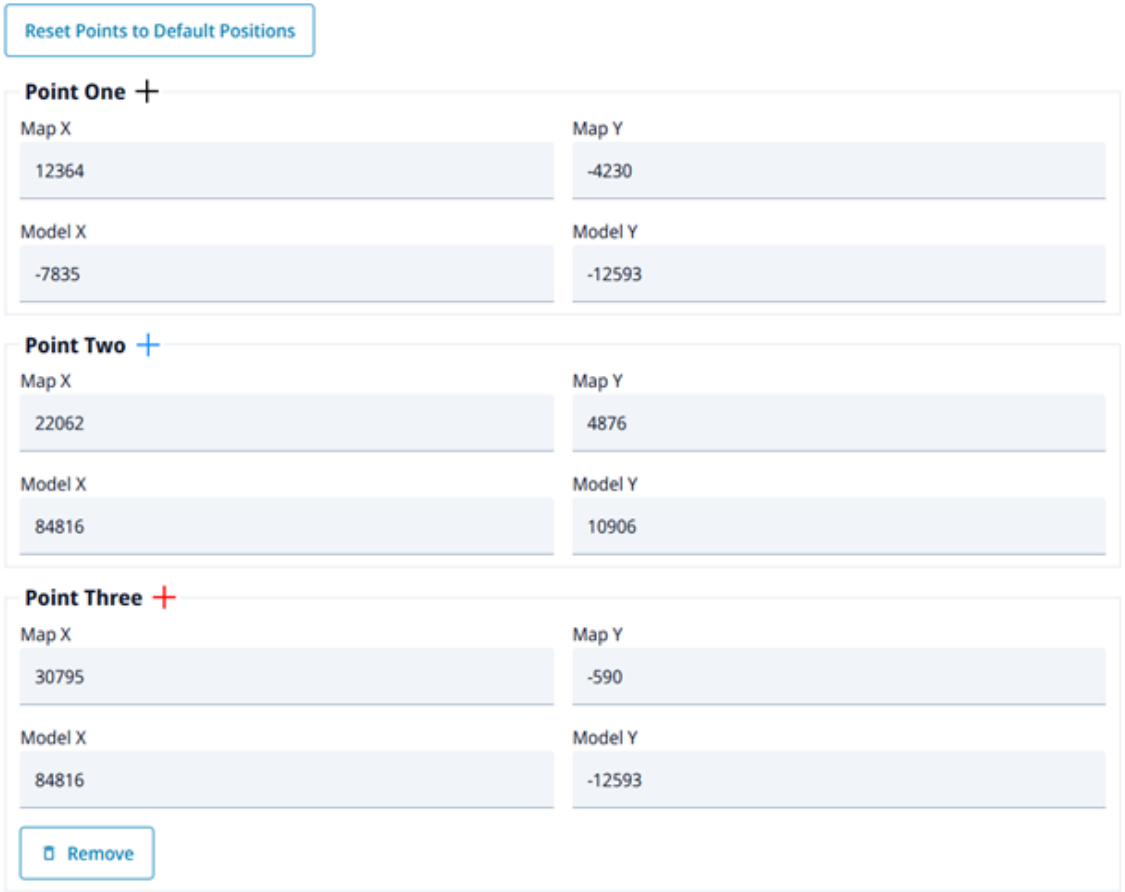


Select two points from the map in the map viewer. Select points that can be easily found in the model. The **Map X** and **Map Y** coordinates will be updated according to the location of the points in the map. You can also specify the points in the **Map X** and **Map Y** fields for **Point One** and **Point Two**. The values must be different from each other.

After the points are set, specify the coordinates of the points to **Model X** and **Model Y** fields for **Point One** and **Point Two**. This will determine the locations the points correspond to in the model. After the points are determined, eShare will scale the map accordingly.

If there is a need to rotate the map in relation to the model, a third point can be added using **Add Third Point**. It can be beneficial if the third point creates a roughly 90 degree

angle between the other two points. The three points should not be on the same line, as this can lead to inaccuracies in the transformation. eShare will warn the user if the three points are almost on the same line.



To remove the third point, click **Remove**.

To reset the point values back to default values, click **Reset Points to Default Positions**.

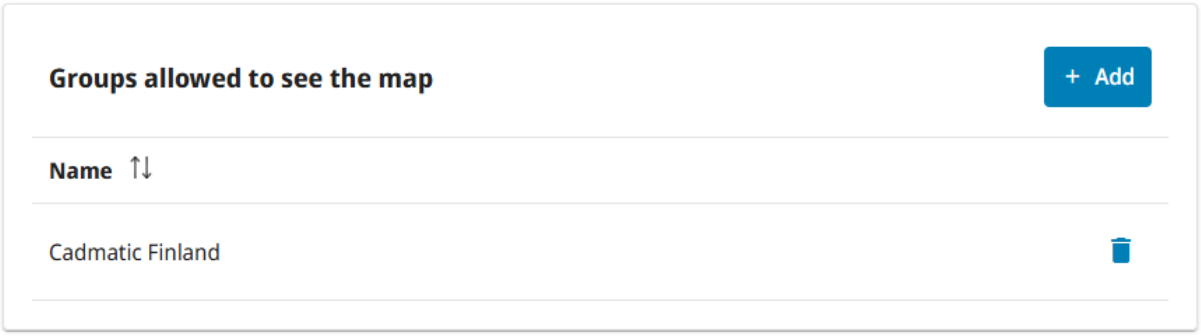
9. Click **Save** to save the map properties.
10. To remove a map, click ⋮ button in Actions column and select **Delete**.The **Remove map** dialog opens. Click **Confirm** to delete the map from the eShare server.

7.5.1. Managing the visibility of a map to user groups

A system administrator can configure in the map properties which user groups are allowed to view a map.

Do the following:

1. Navigate to the project to edit, and then click **Admin** in the main menu. The project administration view opens.
2. Click **Maps**. The **Map Configuration** view opens.
3. Click on the name of the map, or click ⋮ button in Actions column and select **Edit** for the map you want to manage. The Edit map view opens.
4. To add new user groups, select **Add**, select user groups in the Add user groups dialog, then select Add. To delete user groups, select Delete button 🗑 for the user group.



5. Click **Save** to save the map properties.

7.5.2. Replacing an existing map

An existing map in the configuration can be replaced with a new map file. The following file types can be uploaded as a map: DWG, DXF, DGN, JPG, JPEG, PNG, TIF, TIFF, GIF, BMP, and PDF.

Do the following:

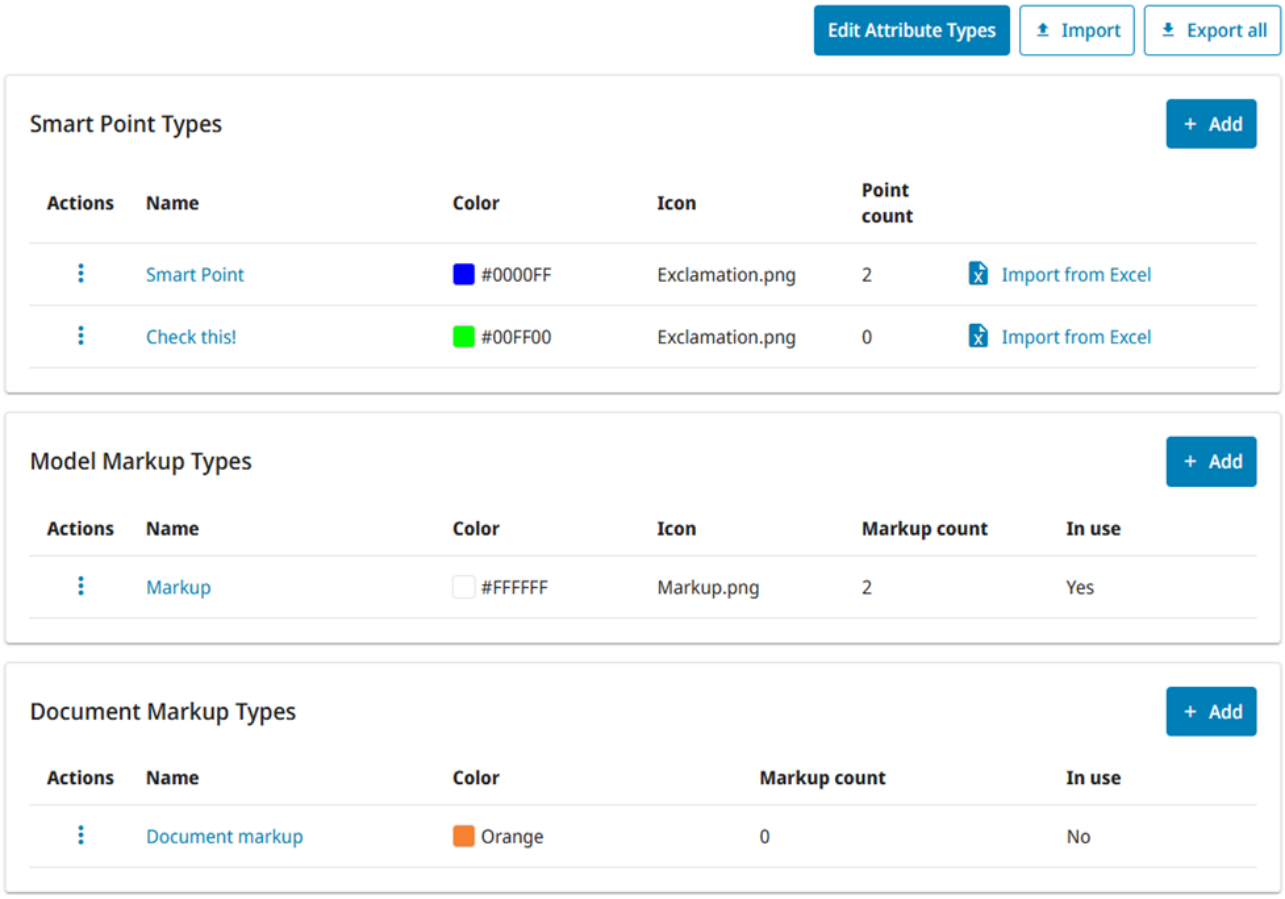
1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Map**. The **Map Configuration** view opens.
3. Click on the name of the map, or ⋮ button in Actions column and select **Edit**.
4. Click **Change Map File**.
5. Select the map file from your computer's files system and click **Open**. A confirmation dialog opens. Click **Change** to replace the existing map with the new map. eShare loads the map.

7.6. Smart Point and markup types

In the **Smart Point and Markup Types** administration view, project administrator can configure Smart Point and Markup Attribute Types, Smart Point Types, Model Markup Types, and Document

Markup Types.

- Smart Point and Markup Attribute Type configuration defines custom attributes that can be used in Smart Point Types, Model Markup Types, and Document Markup Types.
- Smart Point Type configuration defines what kind of Smart Points the users can add to the 3D model.
- Model Markup Type configuration defines what kind of Model Markups the users can add to the 3D model.
- Document Markup Type configuration defines what kind of Document Markups the users can add to the documents.



7.6.1. Exporting and importing the Smart Point and markup type configuration

You can export and import the entire Smart Point and Markup Types configuration (Attribute Types, Smart Point Types, Model Markup Types, Document Markup Types) in text format. You can use this

to copy the settings from one project to another.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Smart Point and Markup Types**. The **Smart Point and Markup Types** configuration view opens.
3. To export the settings, do the following:
    a. Click **Export All**. The **Export Smart Point and Markup Types** text box displays the configuration in text format.
    b. Copy the contents of the text box to the clipboard. You can save the copied settings in a text file if needed.
4. To import the settings, do the following:
    a. Click **Import**. The **Import Smart Point and Markup Types** text box is displayed.
    b. Paste the settings to import into the text box.
    c. **Import default markup properties?** – Select whether to import the properties of the default Markup Type (User Groups, Model Position, Assignee).
        - **Yes** – Import the properties of the default Markup Type.
        - **No** – Keep the current properties of the default Markup Type.
    d. Click **Import**.

7.6.2. Importing Smart Points from Excel

You can import Smart Points directly from an Excel file to the project.

Prerequisites

- You have created the correct Smart Point Type. See Creating a Smart Point type

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Smart Point and Markup Types**. The **Smart Point and Markup Types** configuration view opens.
3. Click **Import from Excel** on the row of the correct Smart Point Type.
4. In the **Import Smart Points from Excel** pane, click **Upload Excel file**.

5. Select the file from your computer's file system and click **Open**.
6. In **Excel File Configuration**, specify the following settings:
    - **Worksheet name** – Specify which worksheet to use, if there are multiple worksheets. The default is to use the first worksheet.
    - **First table row number** – Specify the first table row to be included in the import.
    - **Number of end data rows to skip** – Specify the number of title rows to be skipped.
    - **Has header row** – If you select **Yes**, a header row is not automatically added to the import.
7. Click **Reload and preview file with these settings**. The preview shows the first 5 rows of the Excel file implementing the specified settings.

   The total number of rows is shown under the preview.
8. In **Smart Point Details**, specify the following settings:
    - **Smart Point Type** – Select the correct Smart Point Type.
    - **Name column** – Specify the column with the name of the Smart Point.
    - **External Id column** – Specify the column with the External Id of the Smart Point. Only visible, if the Smart Point Type has an external Id configured.
    - **Overwrite Existing Identical Points** – If you select **Yes**, existing identical points will be overwritten. If you select **No**, the existing identical points will not be overwritten.
    - **Attributes** – Specify the Smart Point attributes. Only visible, if the Smart Point has attributes.
9. In **Coordinates**, specify the following coordinate data.

   Note: The imported coordinate data should be in Project coordinates format.

    - **X coordinate column** – Specify the column containing the X coordinate value.
    - **Y coordinate column** – Specify the column containing the Y coordinate value.
    - **Z coordinate column** – Specify the column containing the Z coordinate value.
    - **Coordinate Scaling Factor** – Specify the scaling factor for the coordinates, if for example feet or meters are used.
10. Click **Test these settings** to test the configuration. The test results, the first 20 Smart Points are shown. The number of encountered errors are shown at the bottom.
11. Click **Import**.

7.6.3. Attribute types

The function of Smart Point and Markup Attribute Types is to allow the user to add information to a Smart Point, Model Markup, or Document Markup. This information can be either plain text that the user enters or a value that the user selects from a predefined list.

7.6.3.1. Creating a Smart Point or markup attribute type

Create the Smart Point or Markup Attribute Types that are needed in the project.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Smart Point and Markup Types**. The **Smart Point and Markup Types** configuration view opens.
3. In the Smart Point and Markup Types configuration view, click **Edit Attribute Types**. The **Smart Point and Markup Attribute Types** view opens.
4. Click **Add Attribute Type**.
5. Enter the Name of the attribute. (You can also change it later, if needed.)
6. Select the Data Type of the attribute:
    - Select **String** if the user is to enter a short text.
    - Select **Multiline** to allow the user to enter a longer text with line breaks.
    - Select **Enumeration** if the user is to select a value from a predefined list, and then click **Add Enumeration Option** to add a list value.
7. Click **Save**. The new attribute type is displayed in the **Smart Point and Markup Attribute Types** list.

7.6.3.2. Editing a Smart Point or markup attribute type

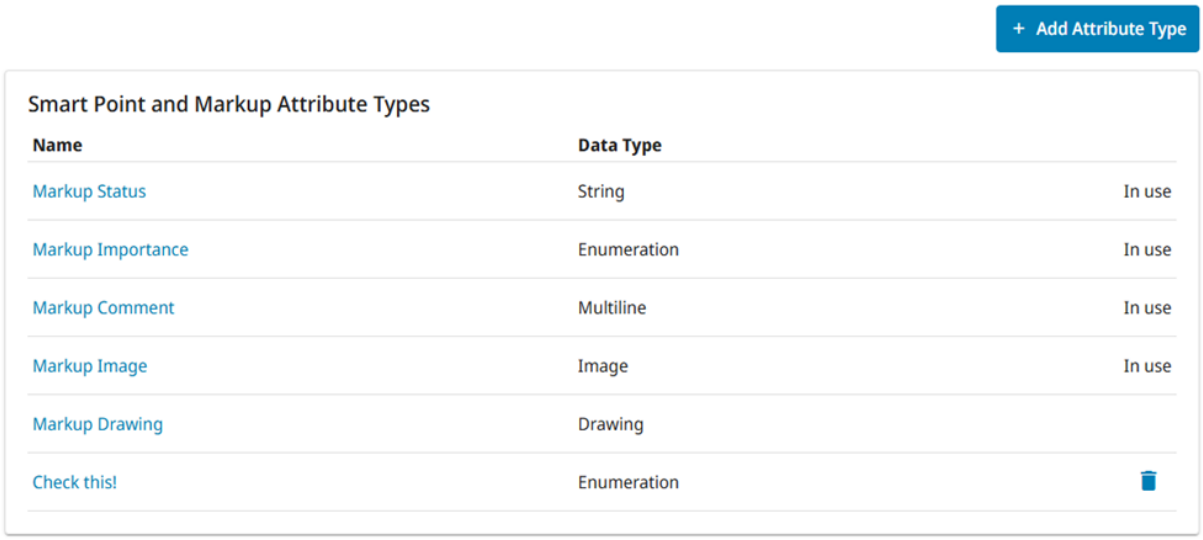
You can always edit the name of an existing Smart Point or Markup Attribute Types. The other properties you can edit as long as the users have not created any Smart Points, Model Markups or Document Markups that use the Attribute Type.

7.6.3.3. Deleting a Smart Point or markup attribute type

In **Project Admin > Smart Point and Markup Types**, click **Edit Attribute Types** to delete any custom Attribute Types that are not in use. If a Smart Point or Markup Type is using the Attribute Type, the

Attribute Type cannot be deleted.

- To delete a custom Attribute Type, click its delete button .



Attribute Types used by the default Model Markup Type ("Markup") cannot be deleted.

Related topics

Smart Point types

Markup types

Exporting and importing the Smart Point and markup type configuration

7.6.4. Smart Point types

Cadmatic eShare and Cadmatic eGo allow the users to add Smart Points to the 3D model.

Smart Points are specialized markers that are linked to an object or object group and display information about that entity. The information that a Smart Point contains can be user-defined or obtained from an external system or database—as defined in the Smart Point Type configuration in eShare. Smart Points can be seen as icons in the 3D model, and the user can select a Smart Point to see the information that it contains. If the Smart Point uses attributes, the user can also add information by editing the attribute values. Smart Points can be created and modified in both eShare and eGo, and the changes can be synchronized from one program to the other. Smart Points can be downloaded also to Cadmatic Plant Modeller. Accordingly, Smart Points can be used to quickly add status information or other comments during a meeting or an on-site visit.

Before users can add Smart Points to the 3D model, project administrator must create the required Smart Point Types. Smart Point Type configuration must be done separately for each project, but a suitable configuration can easily be copied from one project to another.

Once a user has added a Smart Point to the 3D model, the related Smart Point Type can no longer be deleted.

7.6.4.1. Creating a Smart Point type

Create the Smart Point Types that are needed in the project.

Prerequisites

- If the Smart Point Type is to support attributes, first create the relevant attribute types as described in Attribute types.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Smart Point and Markup Types**. The **Smart Point and Markup Types** configuration view opens.
3. In the **Smart Point Types** pane, click **Add**. The **Smart Point Type Details** view opens.
4. In the **Name** field, enter a name for the Smart Point Type.
5. If Smart Points of this type are to be linked to an external data source that provides additional information to the Smart Point, set **External ID** to "Yes". When this is set, a user adding a new Smart Point must provide an external ID that functions as a key to the data source.
6. In **Use Unique Identifiers as References**, select **No**, if the created Smart Point Type is tied to the value of the key attribute of the object or group. Select **Yes**, if unique identifiers are used, and the Smart Point Type is tied to the GUID id of the object or group.

   Note: This functionality should be enabled only in cases when the original 3D model contains GUID data.

   This setting cannot be modified later, if the Smart Point Type is in use.
7. In the **Group permissions for the Smart Point** setting, define the whether users can see, create, modify, and delete Smart Points of this type. Specify the setting for each user group.

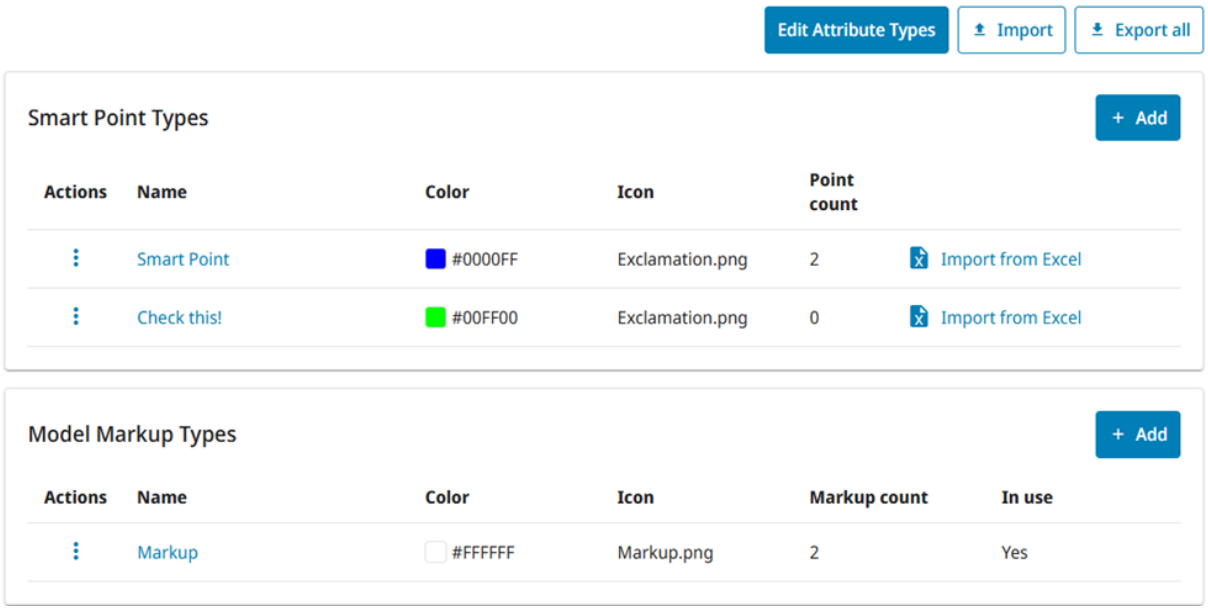
   To add user groups, select **Add**, select user groups in the dialog and then select **Add**. To remove a user group, select Delete button .

8. The **Is Personal** setting, if set to **Yes**, the created Smart Point Type is only visible to the user who creates it and not to other users. The default setting is **No**.
9. The **Display Modification Times** setting is enabled by default. If you do not want Smart Points to display information on creation time, modification time, and users in the 3D model viewer, set this to **No**.
10. Use the **Color** and **Icon** selectors to define how the system highlights an object that has a Smart Point of this type.
11. If Smart Points of this type need to store user-defined information, select the required attribute types from the **Attribute types** list. Define by dragging the order in which the attributes are listed when a user accesses a Smart Point in the 3D model viewer.
12. Click **Save**. The Smart Point Type is displayed in the **Smart Point Types** list, and users can add this type of Smart Points in the 3D model viewer.

7.6.4.2. Deleting a Smart Point type

In **Administration > Smart Point and Markup Types**, in the **Smart Point Types** section you can delete any Smart Point Type.

- To delete a Smart Point Type, click its delete button . A confirmation dialog opens. If the Smart Point Type is in use and there are smart points of the type in use, all the points will also be deleted. Click **Remove** to confirm or **Cancel** to keep the Smart Point Type.



7.6.4.3. Using Smart Points with external data sources

Smart Points can retrieve information from an external data source and display it to the user in the 3D view. To enable this, project administrator must perform the following:

- Set the external data source to provide data for Smart Points. You can use a custom adapter for this, or you can use the built-in Database Adapter and create an Object Attributes Data Source, as described in Creating an object attributes data source.
- Set the Smart Point Type to the type you want it to provide data for. The External ID value of the user-defined Smart Point is used as a Key when looking up data for the Smart Point.

Related topics

Exporting and importing the Smart Point and markup type configuration

7.6.5. Markup types

Cadmatic eShare, Cadmatic eGo and Cadmatic eBrowser allow Markups to be added in the 3D model. Markups typically describe some changes that must be done to a specific part of the model, by showing a screen capture where the user has described the changes with drawing tools and text.

eShare also allows adding Document Markups to documents opened and viewed in eShare. Document Markups typically contain drawings or areas of interest added to the document.

There is one default Model Markup Type, "Markup". Project administrator can create additional Model Markup Types or Document Markup Types as needed.

Once a user has added a Model Markup to the 3D model or a Document Markup to a document, the related Model Markup Type or a Document Markup Type can no longer be deleted.

7.6.5.1. Creating a model markup type

Create the Model Markup Types that are needed in the project.

Prerequisites

- If the Model Markup Type is to support attributes, first create the relevant attribute types as described in Attribute types.

Do the following:

1. Navigate to project that you want to edit, and then click **Project Admin** in the main menu. The project administration view opens.

2. Click **Smart Point and Markup Types**. The **Smart Point and Markup Types** configuration view opens.
3. In the **Model Markup Types** pane, click **Add**. The **Markup Type Details** view opens.
4. In the **Name** field, enter a name for the Model Markup Type.
5. In **Use Unique Identifiers as References**, select **No**, if the Model Markup Type is tied to the value of the key attribute of the object or group. Select **Yes**, if unique identifiers are used, and the Model Markup Type is tied to the GUID id of the object or group.

Note: This functionality should be enabled only in cases when the original 3D model contains GUID data.

This setting cannot be modified later, if the Model Markup Type is in use.

6. In the **Group permissions for the Markup** section, define which user groups are allowed to access Model Markups of this type in the 3D model, on the project's front page, and in search results. For each user group, you can select or remove the following permissions:
   - See
   - Create
   - Modify
   - Delete

Note: System administrators and project administrators are not affected by these settings—they can always access all Markups.

To add user groups, select **Add**, select user groups in the dialog and then select **Add**. To remove a user group, select Delete button .

7. **Color** – Select the color to use for this Model Markup Type.
8. **Icon** – Select the icon to display for this Model Markup Type.
9. **Model Position** – Select how the location of a Model Markup of this type is to be determined in the 3D model.
   - **Click Position** (default) – Use the position that the user clicks to add the Model Markup to the model.
   - **Camera Position** – Use the position where the camera is when the user adds the Model Markup to the model.

Note: The Camera Position option was first introduced in Cadmatic release 2019T3. All Model Markups that have been added in earlier releases use Click Position.

---

10. **Assignee** – Select whether the user can assign a Model Markup of this type to another user when creating or editing the Model Markup.
11. In the **Attribute Types** section lists the attribute types available to this Model Markup Type. You can add an attribute type by selecting it from the field at the end of the list.

    Note: If a user exports Model Markups from eShare and then imports the .ebx file to a Cadmatic product that does not recognize the custom attributes, the attributes are read in as regular Model Markup comments.

12. Click **Save**.

7.6.5.2. Creating a document markup type

Create the Document Markup Types that are needed in the project.

Prerequisites

- If the Document Markup Type is to support attributes, first create the relevant attribute types as described in Attribute types.

Do the following:

1. Navigate to project that you want to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Smart Point and Markup Types**. The **Smart Point and Markup Types** configuration view opens.
3. In the **Document Markup Types** pane, click **Add**. The **Document Markup Type Details** view opens.
4. In the **Name** field, enter a name for the Document Markup Type.
5. **Color** – Select the color to use for this Document Markup Type.
6. In the **Group permissions for the Markup** section, define which user groups are allowed to access Document Markups of this type in the documents, on the project's front page, and in search results. For each user group, you can select or remove the following permissions:
    - See
    - Create
    - Modify
    - Delete

Note: System administrators and project administrators are not affected by these settings—they can always access all Markups.

To add user groups, select **Add**, select user groups in the dialog and then select **Add**. To remove a user group, select Delete button .

7. **Assignee** – Select whether the user can assign a Document Markup of this type to another user when creating or editing the Document Markup.
8. In the **Attribute Types** section lists the attribute types available to this Document Markup Type. By default Document Markups have two mandatory attributes: Markup Image and Markup Drawing. You can add an attribute type by selecting it from the field at the end of the list.
9. Click **Save**.

7.6.5.3. Deleting a markup type

In **Administration > Smart Point and Markup Types**, in the **Model Markup Types** or **Document Markup Types** section you can delete any custom Model or Document Markup Types whose "Markup count" is still 0. If a user has added a Model Markup to the 3D model, or a Document Markup to a document, the related Markup Type can no longer be deleted.

- To delete a custom Markup Type, click its delete button .

"Markup" is the default type of Model Markups and available in all projects—it cannot be deleted.

Related topics

Exporting and importing the Smart Point and markup type configuration

7.7. Point clouds and textured meshes

In the Point Clouds and Textured Meshes configuration view of Cadmatic eShare, a project administrator can upload and remove point clouds and textured meshes, configure and view their status and information, and download generated point cloud files (.cpx).

Note: CPX files generated with version 2023T3 cannot be opened with older versions.

For textured meshes, the supported file formats are FBX, OBJ and GLB files. The texture needs to be either as a linked separate image (with FBX and OBJ files), or an embedded image file (with FBX and GLB files). The supported image file formats are PNG and JPEG.

The mesh files need to be uploaded to eShare as a ZIP file.

As soon as the server has created a .cpx file, it becomes available to eShare App in client computers. You can also use Cadmatic eBrowser or Plant Modeller to open .cpx files. The server automatically deletes any other files from the point cloud folders. The server also deletes .cpx files that no longer have a corresponding source file.

Point clouds always use the project coordinates and are imported there.

Do the following:

1. Navigate to the project, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.

In **Viewing Options** section you can configure the following settings:

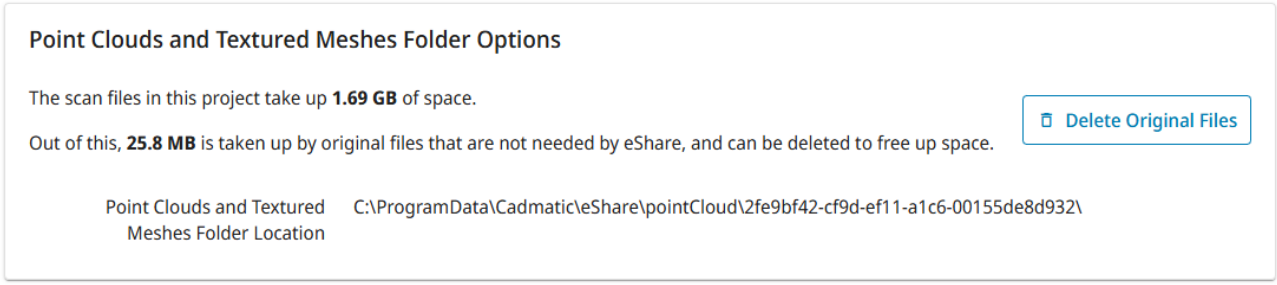
- **Show Point Clouds and Textured Meshes by Default** — Set to **Show** to show point clouds and textured meshes when the model is loaded. Set to **Hide** to hide the point clouds and textured meshes when the model is loaded. The default is **Show**.

  Point clouds and textured meshes set to Hide can still be viewed by toggling their visibility in the model tree in Model view.
- **Color Scale Factor (Brightness)** — Set the color factor value for adjusting the brightness/darkness of the point clouds. Values between 1-10 brighten, and values between 0.001-0.99 darken the point clouds. The default value is 1. This setting only affects point clouds.

7.7.1. Point clouds and textured meshes folder

eShare server automatically converts point cloud files (.e57, .cpd, .cpe, .ptx) into point cloud panorama files (.cpx) and textured meshes (.fbx, .obj) to compressed mesh files (tm7z). **Point Clouds and Textured Meshes Folder Options** shows how much space the point clouds and textured meshes in the project are using. To free up space by removing the original files, click **Delete Original Files**. This will not delete the project's point cloud (.cpx) or textured mesh (tm7z) files., but only the original source files like E57 or FBX files. The behavior of the point cloud folder can be additionally configured using *Cadmatic.RemovePointCloudSourceFilesAfterSync* and *Cadmatic.RemovePointCloudsWithoutSourceFiles* settings in Internet Information Services (IIS) Manager application settings. See Configuring application settings.

Point Clouds and Textured Meshes Folder Options also shows the location of the project's point clouds and textured meshes in the file system, if eShare is running on the same machine as the client.



In eShare server, the location of the project's point clouds and textured meshes is *C:\ProgramData\Cadmatic\eShare\pointCloud* in a folder whose name matches the project ID. The location of the file can be configured using *Cadmatic.PointCloudFolderName* setting in Internet Information Services (IIS) Manager application settings. See Configuring application settings.

eShare server stores source files and the processed point cloud and textured mesh data in this folder. These folders are managed by the server. If uploading large point cloud files through the user interface is problematic, the administrator can directly place the point cloud source files into this folder. Especially for E57 files, if the file should be converted using spherical panorama images from mobile scanning, the file must have the extension .pan.e57. Only insertion of new source files is supported within this folder. All other point cloud and textured mesh administration should be done via eShare admininstration user interface.

7.7.2. Uploading point clouds

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. In **Point Clouds and Textured Meshes Hierarchy**, select the folder for the point cloud files. If you have not added subfolders, only the base folder is available.
4. To add point clouds to the project, click **Upload Files**, or drag and drop the point cloud files there. You can add multiple point cloud files at once.
5. Select the point cloud file or files from your computer's file system and click **Open**.

Note: When uploading E57 files, a dialog with the following prompt **Process E57s as mobile scans or terrestrial scans?** is shown. **Terrestrial (default)** in the correct option when E57 files contain structured point cloud data with scanner position. **Mobile** is the correct option when E57 files contain unstructured mobile scanned point cloud data with spherical panorama images.
If you know that mobile is the correct option, E57 files can also be uploaded directly to the point cloud folder and by changing the file extension from .e57 to .pan.e57.

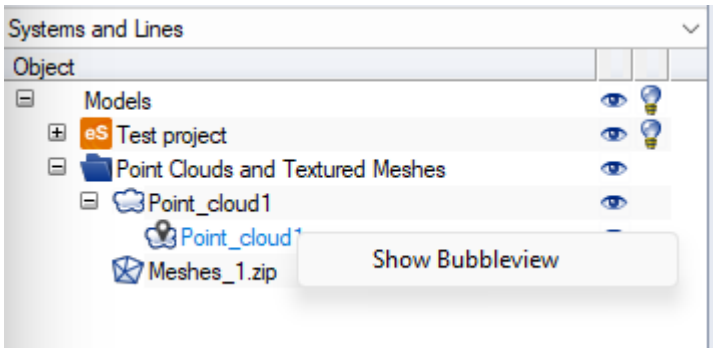
6. In **Transformation Settings** dialog, select **No** if you do not wish to set transformation settings for the point cloud files. Select **Continue**.
   If you select **Yes**, specify the following transformation settings:
   - **Transform / Move** — Set the offset for X, Y, and Z.
   - **Transform / Rotate** — Set the rotation angle in degrees and the coordinates for the X, Y, and Z axes.
   - **Transform / Scale** — Set the scaling factor for adjusting the scale manually.

Note: CPX files cannot be transformed.

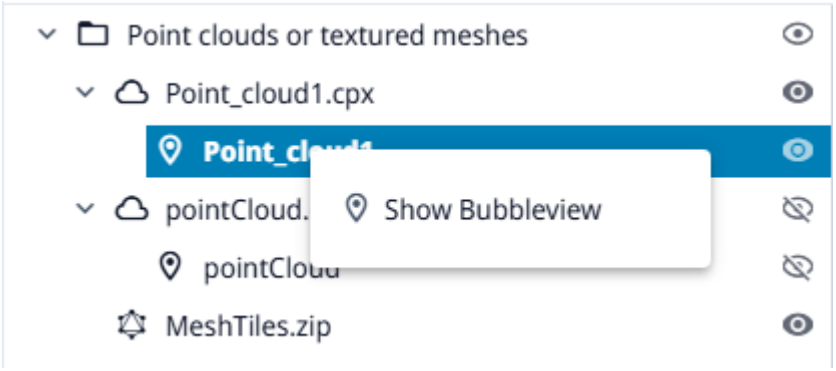
7. Select **Save** and the transformation in completed during the conversion.
   The point clouds are uploaded to the project. A progress bar shows the status of the upload.
   If a transformation was configured, it is saved and can be later viewed in read-only mode, in case future adjustments are needed.

7.7.3. Verifying the display of point clouds

- To verify that the point clouds can be displayed, open eShare App, open the **Model** view of the project, and then in the **Model** tab either double-click a point cloud name or right-click it and select **Show Bubbleview**.



To verify that the point clouds can be displayed, open eShare, open the **Model** view of the project, and then in the **Scans** tab, right-click the name of the point cloud, and select **Show Bubbleview**.



7.7.4. Creating zipped files of textured mesh files

Do the following:

1. Select the folder containing the mesh files from your computer's file system.
2. Right-click the folder and select **Send to > Compressed (zipped) folder**.
   A zipped folder is created and all the files in the folder are added in it.

7.7.5. Uploading textured meshes

Prerequisites

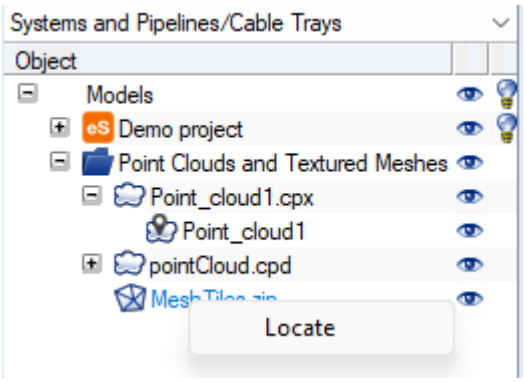
- You have the textured mesh files in a ZIP file.

Do the following:

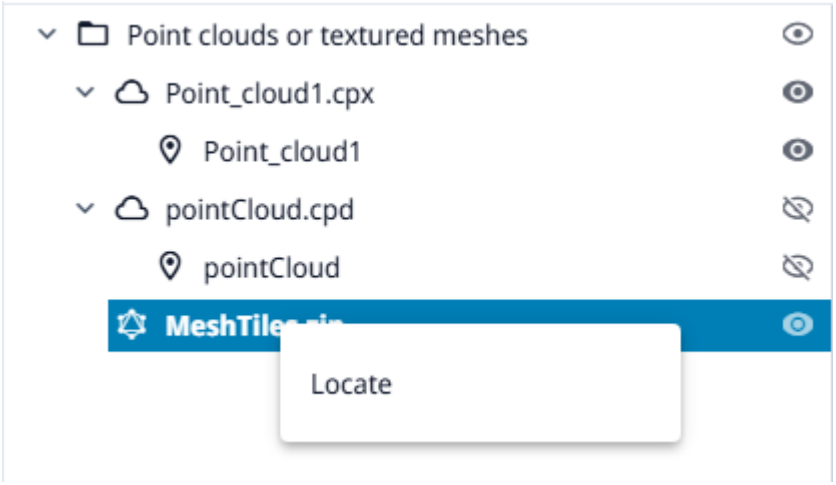
1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. In **Point Clouds and Textured Meshes Hierarchy**, select the folder for the textured mesh ZIP files. If you have not added subfolders, only the base folder is available.
4. To add textured meshes to the project, click **Upload Files**. You can add multiple files at once.
5. Select the textured mesh file or files from your computer's file system and click **Open**.
   The textured meshes are uploaded to the project. The progress of the upload is shown under Status.

7.7.6. Verifying the display of textured meshes

- To verify that the textured meshes can be displayed, open eShare App, open the **Model** view of the project, and then in the **Model** tab either double-click a textured mesh name or right-click it and select **Locate**.



- To verify that the textured meshes can be displayed, open eShare, open the **Model** view of the project, and then in the **Scans** tab right-click the name of the textured mesh, and select **Locate**.

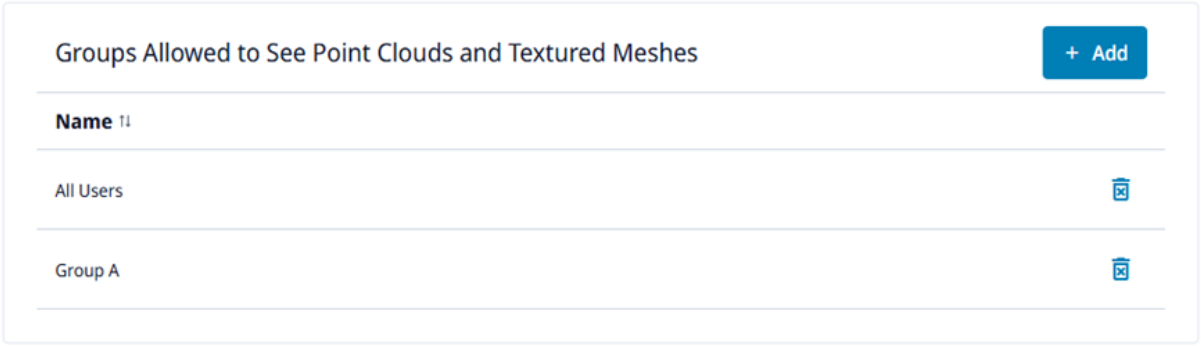


7.7.7. Managing the visibility of point clouds and textured meshes to user groups

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.

2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. In **Groups Allowed to See Point Clouds and Textured Meshes** section you can define which user groups are allowed to see the point clouds and textured meshes.



- If no groups are selected, only administrators will see the data.
- Add user groups with **Add**.
- Remove user groups with the remove button.
- If the **All Users** group is selected, other group selections will be ignored.
- Save changes with **Save**.

7.7.8. Viewing point cloud and textured mesh file details

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. In **Point Cloud and Textured Meshes Hierarchy** section, select the point cloud or texture mesh folder where the file is located. The files in the folder are shown below.
4. Click button for the file, and select **Details**. A dialog with file details opens.

7.7.9. Managing point cloud and textured mesh hierarchies

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.

2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. In **Point Cloud and Textured Meshes Hierarchy** section you can define the hierarchy of point clouds and textured meshes.
4. Click ⋮ button for the point cloud or textured mesh folder for which you want to create a subfolder and select **Add subfolder**.
5. Enter a name for the folder and select **Add**.
6. To rename a subfolder, click ⋮ button and select **Rename**.
7. To delete a subfolder, click ⋮ button and select **Delete**. You cannot delete a subfolder which contains point cloud or textured mesh files.

   Note: You cannot rename or delete the Point Clouds and Textured Meshes base folder.

8. You can also rearrange the folders by dragging.
9. To move files in folders, see Moving point clouds and textured meshes in folders.

7.7.10. Moving point clouds and textured meshes in folders

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. In **Point Cloud and Textured Meshes Hierarchy** section, select the point cloud or textured mesh folder where the files are located. The files in the folder are shown below.
4. Click ⋮ button for the file you wish to move, and select **Move**. You can also move multiple files at once by selecting them and clicking **Move Selected**.
5. Select the new folder from the drop-down menu and select **Save**.
6. You can also move the files by dragging to other folders.

7.7.11. Removing point clouds or textured meshes

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.

2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. Click remove button for the point cloud or textured mesh you want to delete. A confirmation dialog opens. Click **Remove** to delete the file or **Cancel** to keep it.

   To remove multiple point clouds or textured meshes in the folder at once, select them and click **Remove Selected**. A confirmation dialog opens. Click **Remove** or **Cancel** to keep them.

Note: After confirming the deletion of the point clouds or textured meshes, the action cannot be undone.

7.7.12. Exporting and importing point cloud and textured meshes configuration

Note: The exported configuration includes all point cloud, textured mesh, and label detection configurations of the project.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. To export settings, click **Export All**, and then copy the settings from the **Export Point Cloud Options** text box.
4. To import settings, click **Import**, paste the required settings into the **Import** text box, and then click **Import**.

Related topics

AI Label Detection for Point Clouds

7.7.13. AI label detection for point clouds

AI label detection can be used in label detection of point clouds in eShare after it has been enabled and configured by a system administrator. See AI support.

AI label detection can be used in Point Clouds and Textured Meshes configuration view.

7.7.13.1. Detecting labels

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. In **Point Clouds and Textured Meshes** section, select the correct folder, and then all the point cloud files you want to include.
4. Select **Detect labels with AI**. If the detection cannot be started, hovering over the button will display the reason.

   A dialog with the following prompt **Start detection for files? Are you sure you want to start detection for files? This will send your selected files to the selected AI service.** is shown.
5. Select **Start Detection**. Detection begins.
6. To view the progress, select **Detection logs**. The progress for each individual point cloud is shown.
7. To stop detection during the process, select **Stop detection**.
8. After the detection has completed, the following outcomes are shown depending on the results:
   - If new labels are found, click on **New labels to analyze** or **AI Results** to view them. The buttons are disabled until detection is completed in all folders.
   - No new labels found: the view will remain the same as it was before detection was started.
   - If an error occurs while detecting labels, click on the error indicator to view the full error message.

7.7.13.2. Displaying the analysis view of labels

The analysis view allows you to filter, view, and generate Smart Points using the found labels. Analysis view consists of three panes: **Filtering**, **Smart Point Generation Rules**, and the labels themselves.

Prerequisites

- You have completed AI label detection for point clouds and found labels.

7.7.13.2.1. Filtering the labels

In **Filtering Rules** you can filter the found labels. Specify the following:

- **Show** – Select **All labels** to view all labels, or select **Labels that haven't been dismissed or generated**.
- **Fuzzy matching** – Set to **Yes**, if you want to enable matching similar characters. For example 0 and O will be considered a match. When set to **No**, only identical characters are matched.

  Note: This does not apply to regex filters.

- **Case sensitive** – When set to **Yes**, the filter differentiates between uppercase and lowercase letters. When set to **No**, the filter matches words regardless of capitalization.

  Note: This does not apply to regex filters.

7.7.13.2.2. Filtering the results

In Filtering Rules view, there are also filters for the actual results.

In **Label Filtering Rules** section, you can filter the results with five different preset filters:

- **Automatic Detection with Model Keys** – Select the model attributes to use in matching the detection results to the selected attributes.
- **Automatic Detection with Document Object Links**– Select to match the detection results with the already created, indexed links between documents and model objects attributes.
- **Automatic Detection with Document Smart Point Links**– Select the Smart Point types to use in matching the detection results with already created links between documents and Smart Points.
- **Excel file**– Select an Excel file to use. Select the worksheet, column, first table row, and if the document has a header row. Preview the fields from the file by selecting **Preview**. This filter will match the results against the data in the Excel file.
- Filter with regexes where you can define the name of the pattern, and the pattern to be used. This will match the results against the pattern. You can also add a new regex by selecting **Add new**.

  You can also use Cadmatic AI to assist in creating the regex by selecting **Cadmatic AI** in the menu to open chat. See Cadmatic AI.

To save the current filters, select **Save Configuration**. To apply the filters, select **Apply filters**. If there are filtered results, you can select **Reset results** to reset to the original results.

7.7.13.3. Creating Smart Point generation rules

Smart Point Generation Rules pane allows you to select rules for creating Smart Points from the found labels. You can choose if the external ID of the Smart Point should be the label that the AI found, or the matched label from the filters, and which type of Smart Point will be created.

Prerequisites

- You have completed AI label detection for point clouds and found labels.

Do the following:

1. Select **New labels to analyze** after label detection has completed.
2. In **Smart Point Generation Rules** pane, select **Show Options** to expand the pane, and specify the following:
    - **Label to use** – Select **Found Label** or **Matched Label**. If matched label with fuzzy matching is selected, and there are multiple matches to the same label, the matched label with the least changes from the found label will be used.

      Note: This does not apply to labels matched with regex.
    - **Smart Point Type** – Select the Smart Point type to be generated.
3. In **Label Transformation Rules**, select **Add new** to add a new transformation rule.
4. In **Apply to Results From**, select which filters this transformation should apply to, so that you can transform labels from only selected filters if desired. You can use regex named groups to modify the labels to the desired format, similar to other parts of the applications. Named groups can also be created in the regex filters found using the **Label Filtering Rules** and they will be carried over here.
5. To preview the Smart Points to be generated, select **Test transformations and check for duplicates**, if there are transformation rules, or in all other cases select **Check for duplicates**. The results for the first five labels in the results view, or the selected labels if there are any, will be displayed. In **Transformation Results From The Selected Labels**, you can see:
    - original name of the label
    - External Id transformed from the original label based on the rules to be used

- if there already is a Smart Point at the same coordinates with the same External Id
- if there already is a model object with the same External Id.

If the preview is of selected labels, you can remove the label from the selected labels by selecting **Remove from selected**. You can also filter the preview with the duplicate Smart Point, and duplicate External Id columns.

6. Select **Save configuration**.

7.7.13.4. Viewing label detection results

Label detection results pane displays the actual results from the label detection. Depending on the results, different columns are shown. If the results are not filtered, the following will be shown:

- **Label Text** – indicates the label found from the image
- **Confidence** – the confidence of the AI in if the result is correct.
- **Coordinates** – indicates the location of the label in the point cloud.
- **Point Cloud Source** – the name of the point cloud where the label was found
- **Has Identical Smart Point** – indicates if there already is a Smart Point in the same location with the found label name.

If the results are filtered, there will also be columns for

- **Matched Keys** – indicates what data was matched to the label
- **Match** – indicates which filter was matched.

You can filter the results by **Match**, **Confidence**, **Point Cloud Source**, and if the label has identical Smart Points. You can also sort the results according to **Label Text**, **Match**, or **Confidence**.

There also are two buttons between **Label Text** and **Confidence** columns where you can edit the found label, and depending on if the label has multiple matches, view an image or multiple images of the found label in the point cloud, or view all detected label matches from different point clouds with their images on the side. Clicking on the image icon, a dialog opens, which can be resized and moved to a desired position on the screen, and kept open to select new images to be displayed in the dialog. You can zoom in and out of the image, or move the image by dragging. Close the dialog with the close button in the top right corner.

You can select the labels, and either dismiss them by selecting **Dismiss**, or generate Smart Points by selecting **Generate Smart Points**. A warning is displayed, if there are duplicate Smart Points, or items with the same External Id. If you create a large number of Smart Points at once, a confirmation

dialog will be shown. A notification is displayed after the Smart Points have been successfully generated.

To delete labels, select the labels you want to delete, and select **Delete**.

Note: Deleting labels is permanent and cannot be undone after confirming the deletion. If you delete a point cloud the labels that were detected from, the point cloud will have the value *Deleted* in the point cloud source, and the image will be a warning of the point cloud being deleted.

7.7.13.5. Exporting and importing label detection configuration

---

Note: The exported configuration includes all point cloud, textured mesh, and label detection configurations of the project.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Point Clouds and Textured Meshes**. The **Point Clouds and Textured Meshes** configuration view opens.
3. To export settings, click **Export All**, and then copy the settings from the **Export Point Cloud Options** text box.
4. To import settings, click **Import**, paste the required settings into the **Import** text box, and then click **Import**.

7.7.14. eShare point cloud converter

Point cloud files can take up hundreds of gigabytes in their original file formats. To save space, improve downloading point clouds to Cadmatic eShare and, for instance. sharing the point cloud files to other users, the point cloud files can be converted to CPX files. You can use the *PointCloudConverter.exe* tool to convert large point cloud files into CPX files. The supported file types are CPC, CPD, CPE, CPX, E57, and PTX. The tool is located in *\Program Files\Cadmatic\eShareAdminTools\eSharePointCloudConverter*.

Note: CPX files generated with version 2023T3 cannot be opened with older versions.

Run the tool with the required parameters to perform a task:

PointCloudConverter <input>

where **<input>** is the input point cloud file or directory.

If the input is a single file, the output is a single file. If the input is a directory, all files of supported file types in the specified directory will be converted.

To specify the output point cloud file or directory, run the tool with the following:

PointCloudConverter <input> <output>

where **<output>** is the output point cloud file or directory.

To run the tool with options, use the following:

PointCloudConverter <option> <input> [<output>]

The tool supports the following options.

Option	Description
-o --overwrite	Overwrite output files if they already exist
-l --log	Enable logging
--version	Show version information
-i --img	Convert E57 files from panorama images instead of point clouds
-? -h --help	Show help and usage information

7.8. Status tracking and object grouping

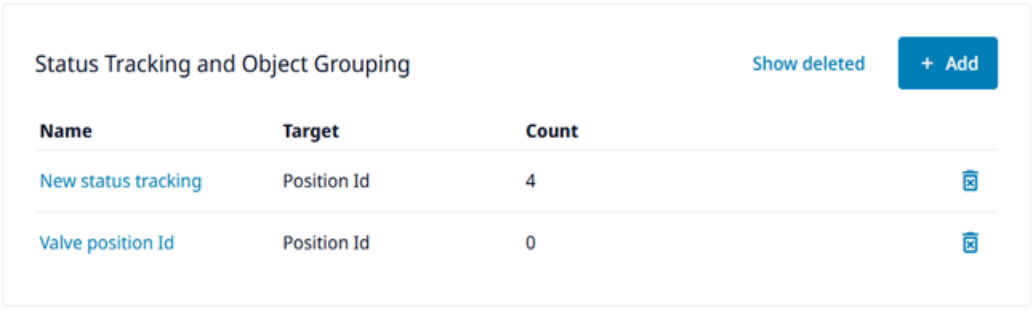
Status tracking allows the various stakeholders to track the status of the design project and to provide feedback to others. That is, when a user is navigating in the 3D model in Cadmatic eShare or

Cadmatic eGo, the 3D view can highlight and categorize objects and object groups based on their status tracking values. Users who have sufficient permissions can change the status values in the properties pane. From search results the status values can be exported to a spreadsheet file.

When a user changes the status of an object in the 3D view, the color of the object changes to match the respective visual style, if the visual style is currently selected to be in use. In object examination mode, however, the colors are not visible. If status tracking is defined for an object group, changing the status of one object in the group will automatically change the status (and color) of all objects in that group.

It is also possible to select multiple objects that have different status values and change their status to a specific common value.

In the **Status Tracking and Object Grouping** settings, project administrator can create the status trackers and tracking values that are needed in the project.



Status tracking types can be removed using button also after they have been taken into use in 3D, but deleting a status tracking type will also delete all the related status trackings. In the confirmation dialog select **Delete status tracking type**.

If there are deleted status tracking types, you can select to show or hide them on the list by selecting **Show deleted** or **Hide selected**.

Related topics

Project users and groups

7.8.1. Creating a new status tracker 112
7.8.2. Modifying existing status trackers 118
7.8.3. Exporting and importing status trackers 118

7.8.1. Creating a new status tracker

Perform the following to create a new status tracker and to define the workflow that users must follow when changing the value. Status trackers always relate to attributes; an object or object group must have the specified attribute for the status tracker to be present. You can create multiple status tracking definitions that relate to the same attribute.

Status tracking can also be used for grouping, if there is need to work with a large number of status values.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Status Tracking and Object Grouping**. The **Status Tracking and Object Grouping** configuration view opens.
3. If there are no existing status tracking attribute types, or you need to create a new one, do the following:
    a. In the Status Tracking and Object Grouping view, click **Edit Attribute Types**.
    b. In the Attribute Types view, click **Add Attribute Type**.
    c. Enter a name for the new attribute and select the data type.
    d. Click **Save**.
4. In the **Status Tracking and Object Grouping** pane, click **Add**. The **Status Tracking Details** view

opens.

5. Define the general settings:
    - **Name** – Enter a suitable name for this status tracker. For example, if the purpose is to track the verification status of parts, the name can be "Part Verified".
    - **Target Attribute** – Select the attribute that an object must have to use this status tracker. Only ID-type (key) attributes and attributes that define a group can be selected. This setting cannot be changed later.
    - **Default Value** – Specify the default value of this status tracker or leave it as "Undefined". You can also select the color to use to visualize objects that have the default value; the default color of these objects is black.
    - **Display Category** – Specify the display name of the status tracking shown in the properties pane. Default is Status Tracking.
    - **Use Unique Identifiers as References** – Specify if the used reference is the key attribute or GUID id of the object or group.

      If **Disabled**, statuses of this type are linked to the value of the Target Attribute of the status the object has. If some objects have identical key attributes, they will share the same status. If **Enabled**, statuses of this type are linked to the GUID id of the object, if the Target Attribute is the key attribute of the object. If the Target Attribute is a group attribute, then the status is linked to the GUID id of the group.

      Note: This functionality should be enabled only in cases when the original 3D model contains GUID data.

    - **Used for Grouping** – If you select **Enabled**, it will affect the following settings:
        - The order of **Status Values** cannot be changed.
        - Project color definitions are not supported for **Status Values**.
        - **Status Tracker Group Permissions** will be limited to See Status, Change Status, and Add New Values.
        - **Status Workflows** section will not be visible and all transitions are allowed.
    - **Users Can Add New Values** – Specify the following:
        - **Enabled** – Select if you want users to be able to add their own status tracking values in the properties pane of the 3D view. You can set the permission separately for each user group in the **Status Tracker Group Permissions** section, but the possible permissions are limited to See Status, Change Status, and Add

New Values.

- **Use Template for New Values** – If adding new values is enabled, you can select to use a template for the new values.

  If set to **Enabled**, any combination of model attributes and text can be entered in the **Value Template** field.

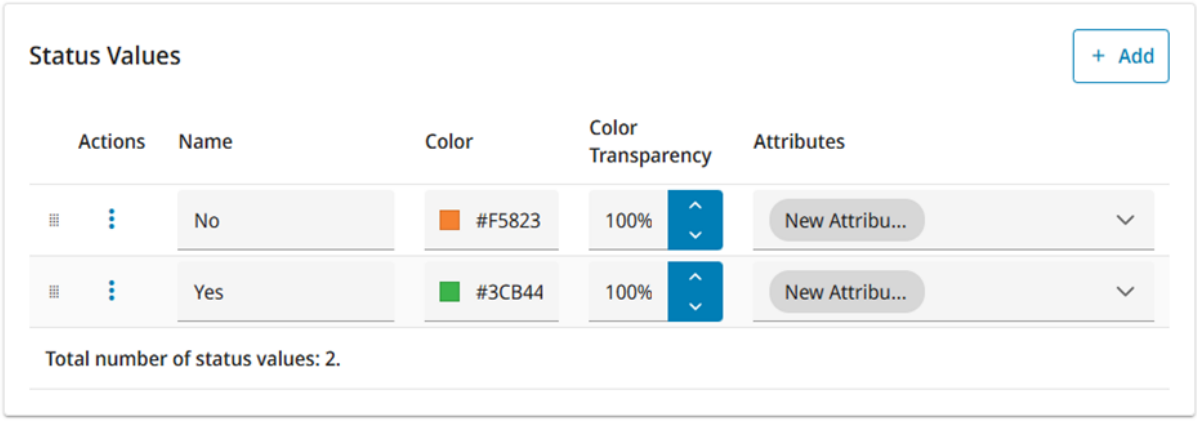
  In the model, when a user clicks on the plus button to start adding a new status tracking value, the attributes in the template are pre-filled in the box, if all the attributes used in the template are found in the selected object. The text can be edited or removed.

- **Disabled** (default) – Select if you want users to be able to only select one of the predefined values.

Note: If you allow users to add values, you cannot define a workflow for the status tracker.

---

6. In the **Attribute Types** section, select the attribute type from the drop-down list.
7. In the **Status Values** section, define the status values that users will be able to select.
   a. To add a value, click **Add**, enter the value name, and select a suitable color. When a user selects this tracking value for an object in the 3D view, the object is shown in the specified color.
   b. When there is more than one value defined, drag the values to designate the order in which the values are listed to the user.

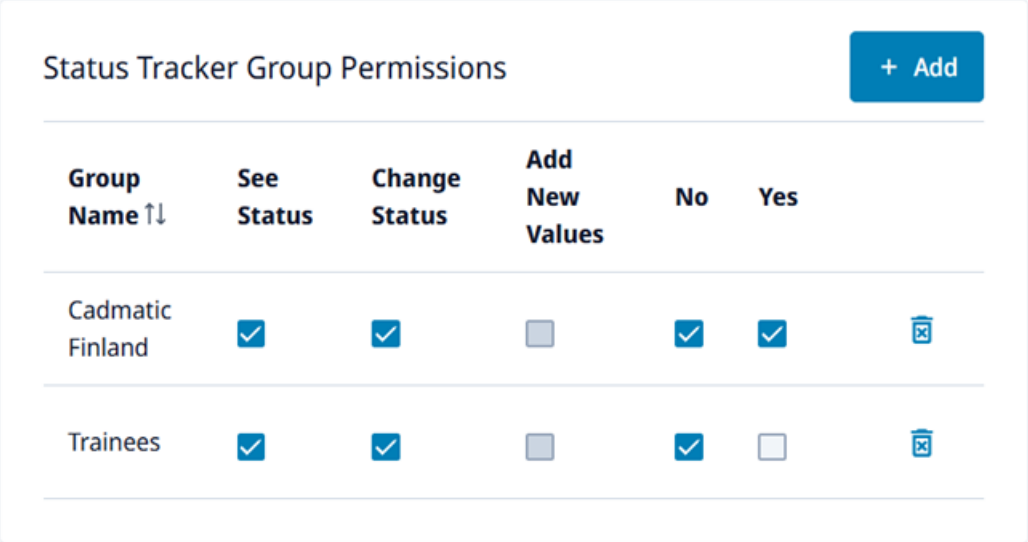


8. In the **4D Sequence** section, if the state is enabled, specify the following:
   - **Excluded object visibility state** – Specifies the visibility of objects, which are not part of the 4D sequence.

    - **Do not change** – The visibility of any model objects without an initial category will not be changed.
    - **Hide all objects** – Every single model object without an initial category is made visible automatically before starting the animation.
    - **Show all objects** – Every single model object without an initial category will be automatically hidden before starting the animation.
- **Step length** – Sets the length of a single step in the 4D sequence. For example, setting the step length as "0 months, 1 days, 0 hours" means the length of each step is 1 day. Setting a low value in this field may result in very long 4D sequences.
- **Start time** – Specify the start time for the sequence. Select from the following options:
    - **From first** – 4D sequence starts from the earliest instance of the data present in the database.
    - **Specific start time** – Select a specific start date from the date picker.
    - **Relative start time** – Set the amount of months, days, and hours to be counted from the time of querying the data to determine the sequence start time. Use negative numbers to indicate past, and positive numbers for the future.
- **End time** – Specify the end time for the sequence. Select from the following options:
    - **To last** – 4D sequence continues until the latest instance of data present in the database.
    - **Specific end time** – Select a specific end date from the date picker.
    - **Relative start time** – Set the amount of months, days, and hours to be counted from the time of querying the data to determine the sequence end time. Use negative numbers to indicate past, and positive numbers for the future.

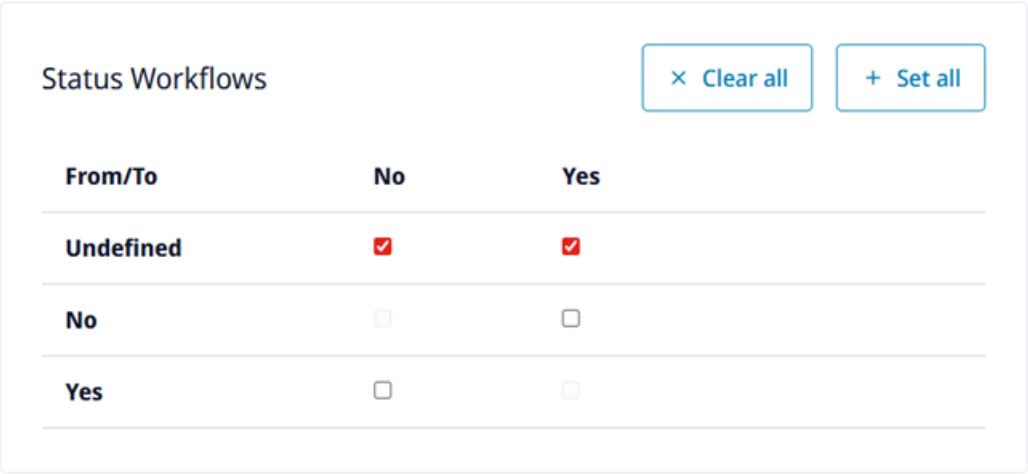
9. In the **Status Tracker Group Permissions** section, define the permissions that each user group has.
    - **Add New Values** – If you enabled **Users Can Add New Values** for this status tracker, select this option to allow the specified user group to add new status values.
    - **See Status** – Select this option to allow the user group to see the status of this status tracker.
    - **Change Status** – Select this option to allow the user group to change the status of this status tracker, either freely or according to the workflow settings.
    - **<status value>** – Select whether the user group can set the status to the specified value.

In this example, both user groups can change the status (one can set it to "Yes" or "No", the other only to "Yes"), and none of the groups can add new status values.



10. In the **Status Workflows** section, specify which status changes will be possible in the user interface. If a status that is listed in the vertical axis (in the **From** column) can be changed to a given status in the horizontal axis (in the **To** row), select the appropriate check box.

    In this example, the user would be able to change the default value "Undefined" to either "No" or "Yes", but not change "No" to "Yes" or "Yes" to "No".



Important: Make sure that Status Tracker Group Permissions do not unnecessarily prevent the user from following the specified workflow.

Note: Once the status has been changed in 3D, the user cannot set it back to the default value.

11. Click **Save**. The settings will be applied to the 3D model after the model is (re-)published.

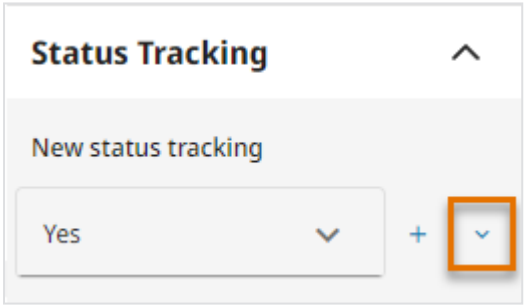
Results

Users can select the status tracker from the visual style menu to see objects in status value specific colors.

Users can select the status tracker from the hierarchy menu to arrange the objects in the model tree according to their status.

When users view or examine an object that has the target attribute specified by the status tracker, the status tracker is displayed in the **Status Tracking** section of the properties pane, and the status can be changed by selecting a value from the list (if allowed by the tracker's configuration). If the target attribute defines an object group, changing the status of one object changes the status for all objects in that group.

After the status has been changed at least once, clicking the small, blue arrow at the right shows a log of the status changes.



Example

If integration to Cadmatic design applications is not in use and you want to track pipe spool statuses, you can create a new attribute whose value is unique for every spool. You can combine the attributes "Idn" (Isometric Drawing) and "Spn" (Spool) in Cadmatic models to a new attribute such as "Ise" whose display name is "Spool Identifier". Define the "Ise" attribute to be of type "Group", and save and republish the model. Then, create status tracking for "Spool Identifier". For details on combining attributes, see Attribute settings.

7.8.2. Modifying existing status trackers

You can modify the settings of a previously created status tracker.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Status Tracking and Object Grouping**. The **Status Tracking and Object Grouping** configuration view opens.
3. On the list of status trackers, click the one you want to edit. The **Status Tracking Details** view opens.
4. You can edit all fields except **Target Attribute**. If the status tracker is in use, you cannot change the setting of **Use Unique Identifiers as References**. You can remove a status tracking value if it is not used by any object.
5. Click **Save**.

7.8.3. Exporting and importing status trackers

You can export and import status tracking settings in text format. You can use this to copy settings from one project to another.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Status Tracking and Object Grouping**. The **Status Tracking and Object Grouping** configuration view opens.
3. To export settings, click **Export All**, and then copy the settings from the **Export Status Tracking** text box.
4. To import settings, click **Import**, paste the required settings into the **Import Status Tracking** text box, and then click **Import**.

7.9. Adapters and data sources

Cadmatic eShare can be integrated with various relational database management systems, file systems, and online services, which allows project-related data to be retrieved from multiple sources.

An administrator must define the required adapters that specify how to connect to a specific target system. Adapter settings are specific to a project, but you can copy settings from one adapter to another by exporting and importing them as text. An administrator can also disable an existing adapter configuration to prevent the system from using that adapter.

Each adapter configuration must have one or more data source configurations that specify what kind of data the adapter should retrieve.

7.9.1. Using attributes and tags in adapters

The objects in a 3D model have attributes that have a human-readable, descriptive *name* and a shorter internal identifier called *tag*. For example, the tag of the Mass attribute is 'mas', and the tag of the Length attribute is 'len'.

Some attributes are *key attributes*—they function as unique identifiers of an object. These attributes are usually referred to as the *position ID* or *position tag* of the object. Models that have been created with Cadmatic Plant/Outfitting typically have key attributes such as Name (nam), Valve Position (vpo), Instrument Position (ipo), Equipment Position (.n5), and Structural Position (.no).

In general, models can contain different kinds of attributes, depending on how the model was created:

- When publishing a model with Plant Modeller, the user can select which attributes to include, and the tag names are as defined by Plant Modeller.
- When using Cadmatic eXchanger to convert a third-party model to the Plant Modeller format, the user can define conversion mappings that determine which attributes and tags will be added to the model.

Adapters that allow Cadmatic eShare to connect to external systems can use tags to specify which attributes trigger the retrieval or generation of additional data. Adapters can use any tags, but usually the tag should identify something specific, and you are not likely to use for example Mass as a key attribute in integration.

A tag does not necessarily identify the object that contains the tag, but a group that the object belongs to, or a stock part that the object refers to. For example, if the tag identifies the part number of a stock part, you can configure an adapter that allows the user to open the relevant part's page from an online parts catalog.

Usually adapters support the following ways of using tags:

- A single tag. This can be a tag from the original 3D model file (.ebm/.ebmx) or a group tag, or the tag of a Derived Attribute.
- A list of tags. You might need to define a list of tags instead of a single tag if the expected data might be found in different tags, depending on the object (for example, in equipment position ID or valve position ID).
- Compound tags. These are attribute combinations that must match the external data source. For example, spools can be identified by the combination of the attributes Isometric Drawing Number (Idn) and Spool Number (spn). Compound tags only support a single data source, and it is recommended to use derived attributes instead. When creating an Cadmatic eShare project from a Cadmatic design application, the default attribute configuration includes an attribute that combines the 'Idn' and 'spn' tags to generate a unique Spool Name attribute.

Tags are case-sensitive, so make sure that you type the tag name correctly.

When providing data for groups, you have two options:

- Use the member tag of the group, for example 'sys' for System. Both the group and its members show the data provided by the adapter.
- Use the group tag of the group, for example '[sys]' for System (Group). Only the group itself shows the data provided by the adapter.

  In both cases, it is assumed that a group has been defined in Attribute Settings.

7.9.2. Adapters and indexing

External documents created by third-party systems and applications might contain references to objects in the 3D model. For example, a text label in a drawing in a project document might contain the unique position ID of an object. These references can be turned into actual links that the user can click to open the object in the 3D view. This automatic linking can be enabled by configuring document handler settings, as described in Document handling.

Cadmatic eShare also has an indexing function that can make the linking bidirectional; indexing allows the user to open a document from the 3D view—from the properties of the 3D object being examined.

Indexing is disabled by default. An administrator can enable indexing for PDF documents that are accessed using the following adapters:

- Database Adapter with Document Data Source
- File System Document Adapter

- REST API Adapter

When indexing is enabled, the system periodically looks for object links in the documents provided by the data source, and if it finds such links it saves the document path and the object link in the document index of the project database.

Modifying a document does not trigger full reindexing but indexing continues in the background without interruptions.

Once documents are indexed, users can access documents from the 3D view as well as search for documents using the search function of Cadmatic eShare.

Before enabling indexing in data source settings, project administrator must define a suitable document handler in the Document Handling view, as described in Document handlers. Links defined in Document Handlers must refer to a key attribute, a group defining attribute, or be unique to be shown in the model.

Links that exist in documents published by a Cadmatic design tool are bidirectional by default, but links that Cadmatic eShare adds to such documents are not indexed and can only be used to jump from a document to a 3D object.

Related topics

Document data source

File system document adapter

M-Files adapter

REST API adapter

7.9.3. Exporting and importing adapter configurations

You can export and import adapter settings in text format. You can use this to copy settings from one project to another, or to back up a configuration before making changes to it.

If you want to store exported settings in a file, make sure to use a text editor that does not add, remove, or change any invisible formatting characters such as tabs and line breaks. If you need to send a configuration by email, use a plain-text file attachment instead of pasting the configuration to the body of the message.

Note: If the configuration contains sensitive data, such as passwords in connection strings, these are displayed in plain text; consider removing any sensitive data before showing a configuration to third parties.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. To export settings, click **Export All**, and then copy the settings from the **Export Adapters** text box.
4. To import settings, click **Import**, paste the required settings into the **Import Adapter(s)** text box, and then click **Import**. If an adapter by the same name already exists, the imported adapter has a number appended to its name. Consider whether you need to edit the imported settings, such as the connection string.

7.9.4. Database adapter

A database adapter provides access to information that is stored in databases or some other data sources that support the SQL query language and a connection type that Cadmatic eShare can use. One database adapter instance only works with one database, but you can create as many instances as needed.

In the database adapter settings, you can define different types of data sources that provide different ways to access the data in the database. For example, you can define an Object Attributes Data Source to enable Cadmatic eShare to show data from a database when a user views an object attribute in the 3D view.

Cadmatic eShare uses a single database user account to access an external database. We recommend that you create a database account with read-only permissions for this purpose.

7.9.4.1. Creating a database adapter

Create a database adapter to enable Cadmatic eShare to connect to an external database and access project data stored in that database.

Prerequisites

- You have the connection string that Cadmatic eShare needs for connecting to the database. For additional information, see Connecting to different types of databases. For a general reference, see http://www.connectionstrings.com.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. Click **Add**.
4. In the Available Adapter Types section, select **Database Adapter**, and then click **Create adapter**.
5. In the Adapter section, enter a name and description (optional) for the adapter, and specify whether the adapter should be enabled.
6. In the Adapter Configuration section, specify the following settings:
    - **Database type** – Select the database type from the list: Microsoft SQL Server, OLE DB, ODBC, or Oracle.

      If you are creating a database adapter for Microsoft SQL Server, select the following:
        - **Connection String Mode** – Select the string mode.
            - If you select **Advanced**, specify the connection string to use for connecting to the database in the **Connection String** field. Select **Test** to check the connection string.
            - If you select **Simple**, you can specify a simple connection by filling and specifying the following information:
                - **Server and instance** – Enter the server and instance in format *<server>\<instance name>*.
                - **Authentication** – Select the authentication method. If you select **Username & Password**, enter them in the fields.

                  You can select **Test** to check the connection string.
                - **Database (optional)** – You can specify the name of the database after successfully testing the connection string.
    - **Connection string** – Specify the connection string to use for connecting to the database.

Note: Cadmatic eShare stores the connection string as plain text, so anyone who can access the project database can see the username and password from there.

Note: Add *Encrypt=True* to connection string in SQL database adapters if you want to use encrypted connection.

7. Click **Save**.

Related Actions

- Creating a 4D sequence data source
- Creating a database adapter data source
- Creating a categorization data source
- Creating a document data source
- Creating a historical data source
- Creating an object attributes data source

7.9.4.2. Connecting to different types of databases

The Database Adapter functionality of Cadmatic eShare has native support for Microsoft SQL Server and Oracle databases. It also supports OLE DB and ODBC that enable connecting to other database types such as Microsoft Access, MySQL, and PostgreSQL. To use OLE DB or ODBC, the appropriate data provider supplied by the database vendor or a third party must be installed on the Cadmatic eShare server computer. Cadmatic eShare only supports 32-bit database providers.

7.9.4.2.1. Microsoft SQL Server

Cadmatic eShare provides native support for Microsoft SQL Server databases, and you just need a suitable connection string to establish connection. Connection must be authenticated using mixed mode authentication; Windows authentication is not supported.

To access a Microsoft SQL Server database, a project administrator must define a database adapter that has database type set to Microsoft SQL Server, and the connection string must be using this format:

data source=<DB server>\<instance name>;initial catalog=<DB name>;User ID=<username>;Password=<password>;MultipleActiveResultSets=True

Note: The database user defined in the connection string only needs to have read access to the database.

Example connection string

This connection string specifies that Microsoft SQL Server database is on the local computer, the instance name is 'MSSQLSERVER', the database name is 'StatusDatabase', and the database is accessed with the username 'eshare' and password 'esharepass':

data source=localhost\MSSQLSERVER;initial catalog=StatusDatabase;User ID=eshare;Password=esharepass;MultipleActiveResultSets=True

---

Note: Add *Encrypt=True* to connection string in SQL database adapters if you want to use encrypted connection.

7.9.4.2.2. Microsoft Access

Cadmatic eShare can connect to Microsoft Access databases (accdb and mdb files) by using Microsoft Office Access Connectivity Engine (referred to as ACE below) or its predecessor Microsoft JetEngine, both of which use an OLE DB connection.

Note: We do not recommend using Microsoft Access databases directly with ACE because its performance is very limited and too slow for most enterprise scenarios, and unexpected behavior can occur when using this type of connection in a server application. A better alternative is to transfer the data from Microsoft Access to a proper database server such as Microsoft SQL Server or Oracle Database.

For Cadmatic eShare to access a Microsoft Access database, the system needs to be prepared as follows:

- The database files are placed in a location that Cadmatic eShare can access.
- Cadmatic eShare has read access to the database files. The simplest way to do this is to grant read access for the IIS_IUSRS group.
- Either Microsoft Access or Microsoft Access Runtime is installed on the Cadmatic eShare server computer. Microsoft Access 2010 Runtime is available for free at http://www.microsoft.com/en-us/download/details.aspx?id=10910.

Then, a project administrator must define a database adapter that has database type set to OLE DB, and the connection string must be using this format:

Provider=<data provider>;Data Source=<database file>

Example connection string

This connection string specifies that Microsoft Access database is located in *C:\Database.accdb* and it does not use password protection:

Provider=Microsoft.ACE.OLEDB.12.0;Data Source=C:\Database.accdb;

To retrieve data from the database, define the required SQL queries in the data source configuration of the database adapter. For example, if the database has a table named 'Table1' with the columns 'Key1' and 'Value1', you can use this kind of query:

SELECT Value1 FROM Table1 WHERE Key1 = @Key

See Creating an object attributes data source for more query examples.

7.9.4.2.3. MySQL

Cadmatic eShare can connect to MySQL databases by using ODBC.

To access a MySQL database, the system needs to be prepared as follows:

- 32-bit version of 'MySQL Connector/ODBC' is installed on the Cadmatic eShare server computer. Version 5.2 or later is recommended, but also earlier versions should work.
- MySQL database provides a user account with permission to access the database; having SELECT privileges should be sufficient.

Then, a project administrator must define a database adapter that has database type set to ODBC, and the connection string must be using this format:

Provider={MySQL ODBC 5.2 Unicode Driver};Server=<DB server>;Database=<database name>;User=<username>;Password=<password>;Option=3;

Note the curly braces {} in *{MySQL ODBC 5.2 Unicode Driver}*. Change this provider name if your site uses a different version of MySQL Connector/ODBC.

Example connection string

This connection string specifies that MySQL database is on the server 'mysql', the database name is 'crmdb', and the database is accessed with the username 'eshare' and password 'esharepass':

Provider={MySQL ODBC 5.2 Unicode Driver};Server=mysql;Database=crmdb;User=eshare;Password=esharepass;Option=3;

7.9.4.2.4. PostgreSQL

Cadmatic eShare can connect to PostgreSQL databases using ODBC connection.

To access a PostgreSQL database, the system needs to be prepared as follows:

- 32-bit version of PostgreSQL ODBC Driver (psqlODBC) is installed on the Cadmatic eShare server.
- PostgreSQL database provides a user account with sufficient permissions; at least CONNECT to the database and SELECT to the tables are needed.

Then, a project administrator must define a database adapter that has database type set to ODBC, and the connection string must be using this format:

> *Driver={PostgreSQL Unicode};Server=<DB server>;Port=<server port>;Database=<database name>;Uid=<username>;Pwd=<password>;*

Note the curly braces {} in *{PostgreSQL Unicode}*. The TCP server port number is typically 5432.

Example connection string

This connection string specifies that PostgreSQL database is on the server 'pgsql' and uses the default port 5432, the database name is 'crmdb', and the database is accessed with the username 'eshare' and password 'esharepass':

> *Driver={PostgreSQL Unicode};Server=pgsql;Port=5432;Database=crmdb;Uid=eshare;Pwd=esharepass;*

7.9.4.3. Creating a database adapter data source

Add a data source to a database adapter to enable the adapter to retrieve specific type of data from the database.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. On the **Adapter**s list, click the database adapter to which to add the data source.
4. In the **Data Sources** section of the adapter settings, click **Add data source**.
5. In the **Available Data Source Types** section, select the data source type to use:

- **Database 4D Sequence Data Source** – To display sequence-based visualization for objects in the 3D view.
- **Database Categorization Data Source** – To display retrieved data in the 3D view using color highlighting or as a hierarchy.
- **Database Document Data Source** – To retrieve documents from the database.
- **Database Historical Data Source** – To display retrieved history data in graph format.
- **Database Object Attributes Data Source**– To display retrieved data in object attributes.

6. Click **Create data source**, and specify the data source specific settings.
7. Click **Save**.

7.9.4.4. 4D sequence data source

4D sequence data source is a data source type which provides a sequence-based visualization for 3D objects in the model view. In essence, this works similarly to categorization-based visual styles, with time added in as a variable. Once configured, 4D sequences are found in the model view the same way as visual styles.

4D sequences are only available in eShare App.

7.9.4.4.1. Creating a 4D sequence data source

Create a 4D sequence data source to enable users to easily visualize progress and changes to objects in 3D view.

Prerequisites

- The project contains a database adapter that can connect to the database that contains the sequence information. See Creating a database adapter if needed.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. On the **Adapters** list, click the database adapter to which to add the data source.
4. In the adapter settings, click **Add data source**, select the data source type **Database 4D Sequence Data Source**, and then click **Create data source**.
5. In the **Data Source** section, specify the following settings:

- **Name** – Enter a name for the data source.
- **Description** (optional) – Enter a description for the data source.
- **State** – Specify whether the data source is disabled or enabled (default).

6. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
   - If no groups are selected, only administrators will see the data.
   - Add user groups with **Add**.
   - Remove user groups with the remove button.
   - If the **All Users** group is selected, other group selections will be ignored.

   The **Adapter Configuration** section is read-only.

7. Specify the **Data Source Configuration** settings:
   - **Excluded object visibility state** – This setting defines what should happen to the visibility of model objects in the 4D sequence which do not have an initial category.
     - **Do not change** (default) – The visibility of any model objects without an initial category will not be changed.
     - **Show all objects** – Every single model object without an initial category is made visible automatically before starting the animation.
     - **Hide all objects** – Every single model object without an initial category will be automatically hidden before starting the animation.
   - **4D step type** – This setting defines what data type is used when determining the steps for the sequence.
     - **Text** – A string-based value from the database is used to determine the names of the steps, which need to be manually added and arranged in the configuration.

       If you select **Text**, see the following:

       Steps section contains a list of all steps that have been added to the configuration. You can use the input field and **Add Step** to manually add step names to the list. If one or multiple queries have been defined in the configuration, you can use **Query for Steps** to find steps by executing database queries defined in the configuration. All found steps can be seen in the drop-down menu below, and you can add steps to the step list by selecting them in the menu, or you can add all of them at once with **Add All Found Steps** button.

- **Time** (default) – A datetime-based value from the database is used to generate the steps.

  If you select **Time**, specify the following:

  - **Step length** – Sets the length of a single step in the 4D sequence. For example, setting the step length as "0 months, 1 days, 0 hours" means the length of each step is 1 day. Setting a low value in this field may result in very long 4D sequences.
  - **Start time** – Sets the time from where the 4D sequence should start.
    - **From first** (default) – 4D sequence starts from the earliest time that the defined queries find from the database.
    - **Specific start time** – 4D sequence starts from the specified date picked with the calendar tool (with time 00:00, meaning that the selected day is included in the 4D sequence).
    - **Relative start time** – 4D sequence starts relative from the time of the user executing the query. This is defined in months, days and hours, which can contain negative and positive numbers. Negative values indicate the past, and positive values indicate the future. For example, setting the relative start time as "-1 months, -14 days, 0 hours" means the start time would be 1 month and 14 days before the time of the query. 0 value in all the fields means the start time is the time of the query ("today").

      Note: Relative start time rounds the given values down: If hours are in use, after calculating the start time it is rounded down to the beginning of the hour (e.g. 1970-01-01 *09:49* is rounded to 1970-01-01 *09:00*). Otherwise, the start time is rounded down to the beginning of the day (e.g. 1970-01-01 *09:49* is rounded to 1970-01-01 *00:00*).

  - **End time** – Sets the time when the 4D sequence should end.
    - **To last** (default) – 4D sequence ends to the latest time that the defined queries find from the database.
    - **Specific end time** – 4D sequence ends to the specified date picked with the calendar tool (with time 23:59, meaning that the selected

day is included in the 4D sequence).

- **Relative end time** – 4D sequence ends relative from the time of the user executing the query. This is defined in months, days and hours, which can contain negative and positive numbers. Negative values indicate the past, and positive values indicate the future. For example, setting the relative end time as "-1 months, -14 days, 0 hours" means the end time would be 1 month and 14 days before the time of the query. 0 value in all the fields means the end time is the time of the query ("today").

  Note: Relative end time rounds the given values up: If hours are in use, the end time is rounded up the end of the hour (e.g. 1970-01-01 *09:49* is rounded to 1970-01-01 *09:59*). Otherwise, the end time is rounded up the end of the day (e.g. 1970-01-01 *09:49* is rounded to 1970-01-01 *23:59:59*).

- **Time offset** – Time offset adjusts the timestamps used for 4D sequencing. eShare assumes all received data uses UTC. If the data does not use UTC, specify the difference in the field in minutes. For example, if the timestamps in the database are saved in UTC+2, set the time offset to "-120".

Note: If you set From first as start time and to last as end time, the animation will use all available data from the query. This may result in a very long and memory consuming animations.

- **Conflict resolution** – This setting specifies the behavior (coloring) of 3D model objects, which return multiple categories in the queries.
  - **First Category** (default) – The object will have the color of the first matching category (ie. topmost in the configuration).
  - **Last Category** – The object will have the color of the last matching category (ie. undermost in the configuration).
- **Queries** – In this section queries to fetch data from the database are defined. The configuration needs at least one query to be operational. Click **Add query** and define the tags and the database query to use:

- In the **Tags** field, specify which tags in the model should match the query results. In the simplest case, the query returns values of a single tag.

  It is also possible to enter multiple tags as a comma-separated list, or use compound tags that combine the values of several tags.

  Do not use group tags in categorization. Instead, use the tag that the members of the group have. For example, use pli instead of [pli].
- In the **Query** field, specify the SQL query to select the columns to be picked for the 4D sequence. Pick the columns in this order:
  - The value used for categorization
  - The time- or text-based value to determine which step the row belongs in
  - As many values as are needed for the Tags section.

  This is an SQL query that returns a number of rows, one row for each model entity for which a category is defined. The query should do a SELECT into the database and return the category as the first column, followed by a column for each tag that is needed to identify the model items.

- **Categories** – Create the categories that should be available to users in the 3D model viewer. You can create a new category manually (click **Add category**) or generate categories by running the database query (click **Refresh**).

  The **Case sensitivity** setting specifies if the value column in the defined categories is expected to be case-sensitive when matching the values to the found values. Case sensitivity does not affect fields that use ranges or regexes (they are always case sensitive).

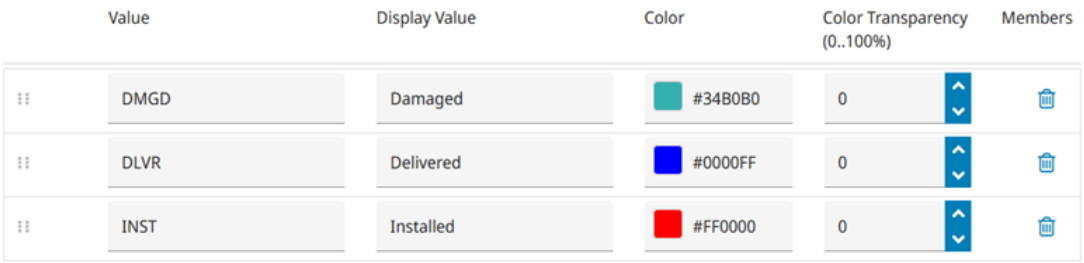
  The **Show to user** setting specifies what categories are displayed to the user in the visual style drop-down menus.
  - If set to **Listed categories**, the user will only see the categories that are defined in this view.
  - If set to **All categories**, the system will create a category for each value that is not defined in this view, and the user will see both manually configured and automatically generated categories. Because these dynamic categories are created on-the-fly and not saved, the color that is assigned to them in visual style might change if the values change.

Note: The categories and member counts here are for ALL the steps in the 4D sequence. All the categories are not necessarily in the same steps, but they are all shown and defined here.

8. Specify the settings of each category:
   a. **Value** – The internal category name. This should match a value that the database query returns. Note that it is also possible that the query returns empty/undefined values.
   b. **Display Value** – This is the name that the users see. If left empty, the Value field is shown to users instead.

      You can define a display name for categories for which the database query returns an empty value for Value field. By default their display name is [Empty], and also null values and values that only consist of white space are handled in the same way.
   c. **Color** – If Visual Style is enabled, use the **Color** tool to pick an appropriate highlight color for categories returned by the query. All categories that are generated by running the query are set to red color by default.
   d. **Color Transparency** – Set the color transparency of the color.
   e. **Members**– Displays the number of matching items in the query result.

   Example category configuration:



Note: If the model contains objects that have no matching tags or objects that do not exist in the database, they will be listed as Uncategorized in the model viewer.

9. You can drag the rows to specify the order in which the categories are to be listed in the model viewer, or click the delete button to delete a category that is not needed.
10. After defined queries have been executed at least once by selecting **Refresh** or **Query for Steps**, the Found 4D Sequencing Data is shown. The table has three columns displaying the 4D-related data found from the database.
    - **Found step** – The name of the step found. These should be in the order of the animation (unless you change the configuration and do not execute the queries again).

If the name is in cursive, the found step is NOT part of the 4D sequence.

- **Number of categories** – Tells the number of different categories that have a change in objects during this step.
- **Number of objects** – Tells the number of objects that change category during this step.

11. Click **Save**.

7.9.4.4.2. Multiple tags in categorization data sources

The database that a categorization data source connects to might contain different types of objects that use a variety of tags to store values that can be seen as belonging to the same category. For example, you might want to create a query that finds all objects that have any kind of position id. In this case, you must define multiple tags for the query, and an object will be included in a category if it has any of those tags and the value of the tag matches the category name.

In the **Tags** field, you define multiple tags by entering each tag on separate line.

For example, if the query is to be matched against the four commonly used position id tags, namely Valve Position (*vpo*), Instrument Position (*ipo*), Equipment Position (*.n5*), and Structural Position (*.no*), the Tags definition would be:

vpo
ipo
.n5
.no

A query that looks for multiple tags basically functions in the same way as a query that looks for a single tag in that the system compares the query results to each tag separately.

7.9.4.4.3. Compound tags in categorization data sources

Some objects in a 3D model cannot be uniquely identified by a single tag, and a combination of tags must be used instead. You can define a compound tag in the **Tags** field of a categorization data source by entering the required tags as a comma-separated list.

For example, piping spools created with Cadmatic Plant Modeller can be identified by the combination of Isometric Drawing Number (*Idn*) and Spool Number (*spn*), and the Tags definition would be:

Idn, spn

A compound tag should only contain tags that are needed to identify the object. All values that the database query returns must match the tag combination, and adding unnecessary tags can have a negative impact on system performance.

You can even include multiple compound tags in a single Tags definition. Each subset must be entered on a separate row and consist of the same number of individual tags.

> For example, if the first compound tag is *Idn, spn*, any additional compound tags must consist of two tags as well. If we add a second compound tag that is based on Pipeline (*pli*) and Spool Number (*spn*), the Tags definition would be:
>
> *Idn, spn*
> *pli, spn*

When using compound tags, the database query needs to refer to one column per included tag, instead of just a single column.

> For example, if Isometric Drawing Number (*Idn*) and Spool Number (*spn*) are used to form a compound tag, the query could be something like this:
>
> Tags: *Idn, spn*
>
> Query: *SELECT Status, ModificationTime, DrawingNumber, SpoolNumber FROM SpoolTable*

7.9.4.5. Categorization data source

Categorization Data Source is a data source type that enables you to use a database query for creating object categories. For example, you could categorize objects according to their construction status (purchased/delivered/installed) or risk level (low/medium/high). When such categorizations exist, the user can select a suitable category from the visual style or hierarchy drop-down menu of the **Model** tab and then view objects that relate to the given category.

7.9.4.5.1. Creating a categorization data source

Create a categorization data source to enable users to easily locate and view objects that relate to a specific category.

Prerequisites

- The project contains a database adapter that can connect to the database that contains the categorization information. See Creating a database adapter if needed.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. On the **Adapters** list, click the database adapter to which to add the data source.
4. In the adapter settings, click **Add data source**, select the data source type **Database Categorization Data Source**, and then click **Create data source**.
5. In the **Data Source** section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
6. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.
    - Remove user groups with the remove button.
    - If the **All Users** group is selected, other group selections will be ignored.

    The **Adapter Configuration** section is read-only.
7. Specify the **Data Source Configuration** settings:
    - **Caching** – This setting defines if data source categorization is always fetched from the data source or if a cached categorization is used.
        - **Disabled** (default) – The data source categorization is always fetched from the data source.
        - **Enabled** – The data source categorization is fetched from the data source when eShare is started or when a data source is configured. After that caching is renewed in intervals defined in **Caching Interval** field. The value is given in minutes.
    - **Visual Style** – When this setting is enabled (default), the user can select the attribute from the visual style drop-down menu, and the 3D view highlights objects with value-specific colors.
        - **Conflict resolution** – This setting specifies which category should override others when multiple categories are defined in this configuration and the database

query returns several matches: **First Category**, **Last Category**, or the special **Multiple Categories** category.

- **Hierarchy** – When this setting is enabled (default), the user can select the attribute from the hierarchy drop-down menu, and the Models tree lists objects in attribute value specific nodes.
    - **Conflict resolution** – This setting specifies which category should override others when multiple categories are defined in this configuration and the database query returns several matches: First Category, Last Category, or the special Multiple Categories category.
- **Queries** – Click **Add query** and define the tags and the database query to use:
    - In the **Tags** field, specify which tags in the model should match the query results. In the simplest case, the query returns values of a single tag that is typically a position id such as Valve Position (vpo) or line id such as Pipeline (pli).
      It is also possible to enter multiple tags as a comma-separated list, or use compound tags that combine the values of several tags.
      Do not use group tags in categorization. Instead, use the tag that the members of the group have. For example, use pli instead of [pli].
    - In the **Query** field, specify the database query. This is an SQL query that returns a number of rows, one row for each model entity for which a category is defined.
      The query should do a SELECT into the database and return the category as the first column, followed by a column for each tag that is needed to identify the model items.

      Example: The database table ValveTable has a **Status** column that we want to use as a category, and it has a column that contains the value of a single model attribute. If the query is providing data for the Valve Position (vpo) tag based on the database column **ValvePosition**, the configuration is:

      Tags – *vpo*

      Query – *SELECT Status, ValvePosition FROM ValveTable*

      This query will return rows where the category name is derived from the **Status** column, and the values in the **ValvePosition** column will be matched against the value of the vpo tag of the objects in the model.

- **Categories** – Create the categories that should be available to users in the 3D model viewer. You can create a new category manually (click **Add category**) or generate categories by running the database query (click **Refresh**).

  The **Case sensitivity** setting specifies if the value column in the defined categories is expected to be case-sensitive when matching the values to the found values. Case sensitivity does not affect fields that use ranges or regexes (they are always case sensitive). For categorizations created in earlier releases, case sensitivity is disabled by default.

  The **Show to user** setting specifies what categories are displayed to the user in the visual style and hierarchy drop-down menus.

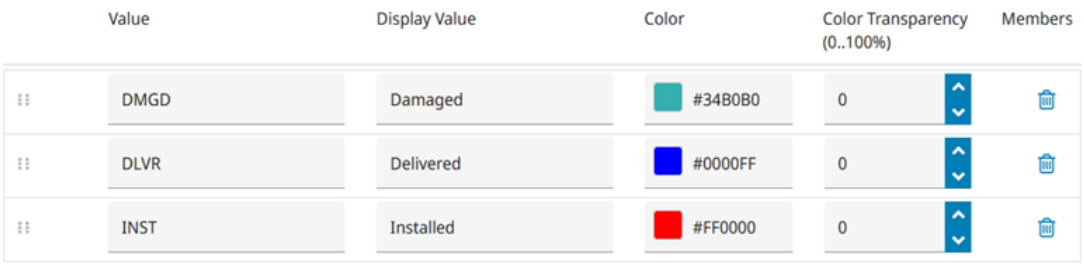
  - If set to **Listed categories**, the user will only see the categories that are defined in this view.
  - If set to **All categories**, the system will create a category for each value that is not defined in this view, and the user will see both manually configured and automatically generated categories. Because these dynamic categories are created on-the-fly and not saved, the color that is assigned to them in visual style might change if the values change.

8. Specify the settings of each category:
   a. **Value** – The internal category name. This should match a value that the database query returns. Note that it is also possible that the query returns empty/undefined values.
   b. **Display Value** – This is the name that the users see. If left empty, the Value field is shown to users instead.

      You can use the same display name for multiple categories, to allow users to view them as a single category.

      You can define a display name for categories for which the database query returns an empty value for Value field. By default their display name is [Empty], and also null values and values that only consist of white space are handled in the same way.
   c. **Color** – If Visual Style is enabled, use the **Color** tool to pick an appropriate highlight color for categories returned by the query. All categories that are generated by running the query are set to red color by default.
   d. **Color Transparency** – Set the color transparency of the color.
   e. **Members**– Displays the number of matching items that according to the query exist in the model.

Example category configuration:



Note: If the model contains objects that have no matching tags or objects that do not exist in the database, they will be listed as Uncategorized in the model viewer.

9. You can drag the rows to specify the order in which the categories are to be listed in the model viewer, or click the delete button to delete a category that is not needed.
10. Click **Save**.

7.9.4.5.2. Multiple tags in categorization data sources

The database that a categorization data source connects to might contain different types of objects that use a variety of tags to store values that can be seen as belonging to the same category. For example, you might want to create a query that finds all objects that have any kind of position id. In this case, you must define multiple tags for the query, and an object will be included in a category if it has any of those tags and the value of the tag matches the category name.

In the **Tags** field, you define multiple tags by entering each tag on separate line.

For example, if the query is to be matched against the four commonly used position id tags, namely Valve Position (*vpo*), Instrument Position (*ipo*), Equipment Position (*.n5*), and Structural Position (*.no*), the Tags definition would be:

vpo
ipo
.n5
.no

A query that looks for multiple tags basically functions in the same way as a query that looks for a single tag in that the system compares the query results to each tag separately.

7.9.4.5.3. Compound tags in categorization data sources

Some objects in a 3D model cannot be uniquely identified by a single tag, and a combination of tags must be used instead. You can define a compound tag in the **Tags** field of a categorization data source by entering the required tags as a comma-separated list.

> For example, piping spools created with Cadmatic Plant Modeller can be identified by the combination of Isometric Drawing Number (*Idn*) and Spool Number (*spn*), and the Tags definition would be:
>
> *Idn, spn*

A compound tag should only contain tags that are needed to identify the object. All values that the database query returns must match the tag combination, and adding unnecessary tags can have a negative impact on system performance.

You can even include multiple compound tags in a single Tags definition. Each subset must be entered on a separate row and consist of the same number of individual tags.

> For example, if the first compound tag is *Idn, spn*, any additional compound tags must consist of two tags as well. If we add a second compound tag that is based on Pipeline (*pli*) and Spool Number (*spn*), the Tags definition would be:
>
> *Idn, spn*
> *pli, spn*

When using compound tags, the database query needs to refer to one column per included tag, instead of just a single column.

> For example, if Isometric Drawing Number (*Idn*) and Spool Number (*spn*) are used to form a compound tag, the query could be something like this:
>
> Tags: *Idn, spn*
>
> Query: *SELECT Status, DrawingNumber, SpoolNumber FROM SpoolTable*

7.9.4.5.4. Categorization data source and hierarchy levels

When using a categorization data source, the top level that the user sees in the hierarchy drop-down menu of the **Models** tab is always a category name. At lower levels the menu items represent

the tags that the queries use; the second level in the hierarchy is based on the first tags in the queries, and so on.

If only a single tag or multiple single tags are used, the objects are listed directly under the categories. If compound tags are used, the first tags in each compound tag represent the second hierarchy level, the second tags represent the third hierarchy level, and so on. If the same tag occurs multiple times, only the first occurrence is used as a hierarchy level.

> For example, if the query uses Isometric Drawing Number and Spool Number, the hierarchy will have the category as the first level, Isometric Drawing Number as the second level, and Spool Number as the third level.

7.9.4.6. Document data source

Cadmatic eShare supports a couple of ways of accessing documents stored in a file system. A database adapter can function as a general-purpose document management adapter when document data is available in a certain way:

- Document metadata, including the hierarchical folder structure, is stored in a document database.
- Documents are stored in a file system that Cadmatic eShare can access, and the document database points to those files.
- Each document consists of exactly one PDF file.
- To enable the system to inject 3D object links into the documents, the PDF files must be searchable, and text must be stored as text, not as graphics.

For example, you could utilize this functionality by exporting data from a document management system and configuring a database adapter to read the exported data from a specified location.

7.9.4.6.1. Creating a document data source

Add a document data source to a database adapter to enable the adapter to provide access to PDF documents and to enable objects in the 3D model to be automatically linked to documents.

Prerequisites

- A database adapter that connects to the document database. See Creating a database adapter.
- To enable indexing, a suitable document handler must be defined. See Adapters and indexing.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. On the **Adapters** list, click the database adapter to which to add a data source.
4. In the **Data Sources** section of the adapter settings, click **Add data source**.
5. In the **Available Data Source Types** section, select **Database Document Data Source**, and click **Create data source**.
6. In the **Data Source** section, specify the following settings:
   - **Name** – Enter a name for the data source.
   - **Description** (optional) – Enter a description for the data source.
   - **State** – Specify whether the data source is disabled or enabled (default).
7. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
   - If no groups are selected, only administrators will see the data.
   - Add user groups with **Add**.
   - Remove user groups with the remove button.
   - If the **All Users** group is selected, other group selections will be ignored.

   The **Adapter Configuration** section is read-only.
8. In the **Data Source Configuration** section, specify the following settings. The different queries that are defined here require unique document identifiers. It is up to you to decide what to use as document ID; typically, it is an internal identifier used in your database.
   - **Indexing** – Select if document indexing is enabled. If you enable it, the system can search for object links in documents, store those links in the project database, and use this document index to create links from 3D objects to relevant documents. The documents must be found in the subfolders of the document base path.

     Note: If you save the data source configuration with indexing, all data sources of the same adapter with indexing enabled, will be indexed again.

     If **Enabled**, specify the following settings:

- **Indexing Speed**– Define the duration of indexing.
  If you select **Duration**, specify the target duration of each indexing round in **Target Indexing Duration** field (in minutes). The set duration takes effect only after the first full round of indexing has been completed. The default is 120.
  If you select **As Fast As Possible**, the target of each indexing round is to be completed in the shortest time as possible.
- **Indexing Mode**– Indexing is done every time the data source configuration is saved. Indexing can also be triggered manually from the Project Administration's General view.
  If you select **Once**, indexing is done only after saving.
  If you select **Daily**, indexing is done once every day.
  If you select **Once per Days of Week**, indexing is done once on the selected days.
  If you select **Given Times in Days of Week**, indexing starts on given times of the day on the selected days. The times are given in the server's local time.
  If you select **Days of Month**, indexing is done once on the selected days of the month.
  If you select **All the Time**, indexing is done continuously. Warning: This can consume a lot of resources if As Fast as Possible is selected as indexing speed.

- **Load Subfolders On Demand**— If enabled, the entire document tree is not loaded at once, but the contents are loaded and visible following the user's examination. The default is **Disabled**.

The following Displaying Documents settings specify the document base path in the file system and the document metadata fields that will be available. For more information and query examples, see Displaying documents.

- **Load Documents From** — Select if the documents are retrieved from **File System** or **Database**. The default is file system.
- **Document base path** – This setting specifies the base location under which documents are stored in the file system. Visible when **Load Documents From** is set to **File System**.
- **Document File Query** – Enter a database query that provides document metadata. When documents are loaded from the file system, at the minimum, the query must return the file name (including the path, relative to document base path) and document's intended display name. Additional fields that it returns will be displayed as document attributes in the **Documents** view.

When documents are loaded from the database, at the minimum, the query must return the document binary blob and document's intended display name. Additional fields that it returns will be displayed as document attributes in the **Documents** view.

Note: When retrieving documents directly from the database, the selected name of the document must include its file extension.

- **Test Document ID** – Enter a document ID and click **Test Query** to check that the query returns expected results.

The following Folder Structure settings allow the document folder structure to be browsed and the documents in it to be opened from the Documents view of Cadmatic eShare. For more information and query examples, see Listing all available documents on the documents tab.

- **Query for Root Folders** – Enter a database query that defines the top-level folders under the document base path. At minimum, the query must return a unique folder identifier and a folder name. Click **Test Query** to check that the query returns expected results.
- **Query for Subfolders** – Enter a database query that defines the subfolders under the root folders. At minimum, the query must return a unique folder identifier and a folder name.
- **Test parent ID** – Enter an ID of a root folder and click **Test Query** to check that the query returns expected results.
- **Query for Documents in Folder** – Enter a database query that defines which documents are contained in each folder. At minimum, the query must return a unique document identifier and a document name.
- **Test folder ID** –Enter a root folder or subfolder ID and click **Test Query** to check that the query returns expected results.

The following Locating Documents settings enable documents to be associated with objects in the 3D model, using document metadata stored in the document database, and to be searchable with simple search.

- **Link Queries** – Click **Add Link Query** to define serving tags and database query to use for locating links. For more information and query examples, see Accessing documents from model items.

- **Search Query** – Enter the database query to use when searching document files with the simple search of Cadmatic eShare. For more information, see Searching.
- **Test Search Term** – Enter a search term and click **Test Query** to check that the search query returns expected results.

Results

Depending on which settings you defined, some or all of the following actions will be possible:

- The document tree and the documents it contains can be browsed in the **Documents** view.
- Simple search can find documents from the document tree.
- The system can associate objects in the 3D model with matching documents.

7.9.4.6.2. Displaying documents

To be able to read the document content, the adapter needs two settings: document file base path and document file query.

Document base path is the root file path where the documents provided by this data source reside, and it must be accessible to Cadmatic eShare server to be used. A trailing backslash for the directory is not necessary.

Document base path example:

C:\Documents\ExampleProject

The document file query provides the document's metadata (attributes) as well as the relative path of its file. Use the Document Identifier provided by a Link Query, a Search Query, or a Query for Documents in Folder as the key of the query. Return the name of the file, including the path relative to the base path, as the first column in the query. Return the document name shown to the user as the second column in the query. If your query returns more columns, these are used as metadata shown to users in the documents view, and can be used to define links in different document handlers.

Document file query example:

SELECT Filename, Name FROM Documents WHERE Id = @Key

@Key will be replaced with the value of the document identifier for the document that triggered the query.

7.9.4.6.3. Listing all available documents on the documents tab

In the Documents view, users can browse documents using either a tree view or list view. In order to populate this view, a project administrator must define three queries in the Document data source:

1. A query for Root Folders.
2. A query for Subfolders.
3. A query for Documents in Folder.

Together these three queries define a tree structure of documents that users can browse.

Query for root folders

You can define all the top-level (root) folders relative to the Document base path using this query. The root folder query must return two columns for each folder. The first column defines a unique identifier for the folder. The second query defines a display name for the folder. The users will see the name of the folder in the Documents view.

Root folder query example:

```
SELECT Id AS FolderId, Name AS FolderName
FROM DocumentFolders WHERE ParentId IS NULL
```

You can verify that the root folder query is working as you intended by clicking the **Test Query** button under the root query.

Note that folder identifiers do not need to be integers: you can also use the folder paths as the identifiers, for example. The only requirement is that the identifiers are unique within the document data source.

Handling folder paths in SQL without a table specifically for the folders can be awkward. For example, if you have a single table with relative file names in it, the root folders can be queried with:

```
SELECT DISTINCT
SUBSTRING([File], 1, CHARINDEX('\', [File])) AS FolderId,
SUBSTRING([File], 1, CHARINDEX('\', [File])-1) AS FolderName
FROM [TableName]
```

Due to the complexity of this and the other queries described below, it is recommended that you include a second table in your database to describe the folder structure, if possible.

Query for subfolders

Use this query to define the subfolders of a given folder. The given folder identifier can be that of a top-level folder or a subfolder. The subfolder query must return two columns for each subfolder. The first column defines a unique identifier for the subfolder. The second column defines the name of the subfolder. The users will see the name of the subfolder in the **Documents** view.

Subfolder query example:

```
SELECT Id AS FolderId, Name AS FolderName
FROM DocumentFolders WHERE ParentId = @Key
```

You can test the subfolder query by providing a parent folder identifier and clicking the **Test Query** button under the subfolder query. You can find a valid parent identifier by testing the root folder query.

If the specified folder does not have any subfolders, the query should not produce any results.

Note that a subfolder query is required even if your hierarchy only contains a single level. In this case, you can use a query like:

```
SELECT Id AS FolderId, Name AS FolderName
FROM DocumentFolders WHERE 1 = 0
```

In the example above with relative file names in a single table, subfolders of a given folder can be queried with:

```
SELECT DISTINCT
(@Key + SUBSTRING(SUBSTRING([File],LEN(@Key)+1,100000),1,CHARINDEX('\', SUBSTRING
([File], LEN(@Key)+1, 100000)))) AS FolderId,
(SUBSTRING(SUBSTRING([File],LEN(@Key)+1,100000),1,CHARINDEX('\', SUBSTRING([File], LEN
(@Key)+1, 100000)))) AS FolderName
FROM [TableName]
WHERE SUBSTRING([File], 1, LEN(@Key)) = @Key
AND CHARINDEX('\', SUBSTRING([File], LEN(@Key)+1, 100000)) > 0
```

Query for documents in folder

Use this query to define which documents are contained in each folder. The given folder identifier can be that of a top-level folder or subfolder. The document query must return two columns for each file. The first column returns the identifier of the document. The second column returns the name of the document. The name is displayed in the Documents view.

Note that you can combine several columns into one result column, for example to display both the drawing number and title as the document name.

Document query example:

```
SELECT Id AS DocumentId, DrawingNr + ' ' + Title AS Name
FROM Documents WHERE FolderId = @Key
```

When defining a document query, you can test it by providing a folder identifier and clicking the Test Query button. You can find a valid folder identifier by testing either the root folder query or the subfolder query.

Again referring to the example above with only relative file names in the database, documents in a given folder can be queried with:

```
SELECT Id AS DocumentId, Title AS Name
FROM [TableName]
WHERE SUBSTRING([File], 1, LEN(@Key)) = @Key
AND CHARINDEX('\', SUBSTRING([File], LEN(@Key) + 1, 10000)) <= 0
```

7.9.4.6.4. Accessing documents from model items

Link queries answer the question “Which documents are associated with model item X?”. This mapping must be based on metadata in your document database. For example, documents can have a field ‘System’ that contains a system name which can be found in the model tag ‘sys’.

For each query, define the model tags that are using the query by adding the tags to the Serving Tags field, one tag per line. Then write the query to map the tag value to a document. Return the identifier of the document as the first column of the result. You can return the name of the document as the second column of the result, but this is optional.

Link query example:

```
SELECT Id AS DocumentId, Name AS DocumentName
FROM Documents WHERE SystemName = @Key
```

7.9.4.6.5. Searching

If you want to search for documents from a document data source, use the Search Query to return document identifiers based on a search string. The first column that the query returns is the document identifier, and any subsequent columns are ignored. You can use this query through the simple search user interface.

Search query example:

SELECT Id AS DocumentId FROM Documents WHERE Name LIKE '%' + @Key +'%'

7.9.4.7. Historical data source

A historical data source retrieves history data for objects or groups and displays it as a history graph in the properties pane of the object or group.

7.9.4.7.1. Creating a historical data source

Create a historical data source to enable history data to be displayed as a graph.

Prerequisites

- To create a data source for Smart Points, the required Smart Point Type has been created as described in Creating a Smart Point type.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. On the **Adapters** list, click the database adapter to which to add the data source.
4. In the adapter settings, click **Add data source**, select the data source type **Database Historical Data Source**, and then click **Create data source**.
5. In the **Data Source** section, specify the following settings:
   - **Name** – Enter a name for the data source.
   - **Description** (optional) – Enter a description for the data source.
   - **State** – Specify whether the data source is disabled or enabled (default).
6. In the **Allowed for User Groups** section, select the **Is Allowed** check box of the user groups that should be allowed to see the data that this data source provides.

- If no groups are selected, only administrators will see the data.
- If the **All Users** group is selected, other group selections will be ignored.

The **Adapter Configuration** section is read-only.

7. In the **Data Source Configuration** section, specify the following settings:
   - **Provide data for** – Select whether to retrieve the data to objects or Smart Points.
     - **Model Objects** – the retrieved data will be added to objects that have a specific attribute.
     - **Smart Points** – the retrieved data will be added to Smart Points of the specified type.
   - **Parameter Set**– Enter the attributes for which the data source is to provide data. There can be multiple attributes in a parameter set, and multiple parameter sets. To add another parameter set, click **Add Alternative Parameter Set**. This option is visible if you set **Provide data for** to **Model Objects**.
   - **Smart Point type served**– Select the appropriate Smart Point from the list. This option is visible if you set **Provide data for** to **Smart Points**.
   - **SQL Query** – Enter the SQL query to use to retrieve data.
   - **Test Key** – There will be multiple fields, if multiple parameter keys are used in the query.
8. Click **Test Query** to test the configuration.
9. In **Columns** section, specify the following:
   - **Time Column** – Select which column's values are used as the time value for the graph.
   - **Data Column** – Select which column's values to use as the data value for the graph.
   - **Refresh Automatically** – **Disabled** by default. If set to **Enabled**, the attributes are refreshed every few seconds. Set the **Refresh Interval** in seconds. Default is 10 seconds.
10. In **Graph Options** section, specify the following:
    - **Is Enum** – If set to **Yes**, and data in the configured data column is enumerated type, the **Detected Enum Values** table will list the distinct values found in the data column. The fetching and detecting of these values is not done automatically and the Refresh button has to be pressed every time a new type of enum values appear or older ones disappear from the data column.

      The data in the Detected Enum Values table can be modified by dragging the values to the desired level. The enum values will be presented in the same order in the resulting

graph's y-axis as they are in the table. Meaning that the top most value in the table will be the highest value in the graph and the last value in the table will be the lowest value in the graph.

- **Unit**(optional) – You can specify a data unit for the graph.
- **Description**(optional) – You can specify a description for the graph.
- **Display Name**(optional) – You can specify a custom display name for the graph.
- **Display Category** (optional) – You can specify a custom display category for the graph.
- **Is Date Local in DB** – If date in database is local, set as **Enabled**.

11. Click **Save**.

Selecting attributes

When a user working with a project chooses to see the details of an object in the 3D model, Cadmatic eShare looks up the attributes the object has and makes data queries for those attributes. If a data source has any of these attributes in **Parameter Set** configuration, then the data source is queried with the values of the attributes from the model object, and the results are shown to the user.

> Example: The model contains an object with the attribute *Valve Position ID* having a value of 12345. A data source has **Parameter Set** set to *Valve Position ID*. When the user asks for details of the object, the data source will perform its query with a key value of 12345 and returns attributes if the database contains data for this key.

Multiple parameter sets are useful if there are several attributes that identify items of the same general kind. For example, you might want to use the same data source for equipment and valves if the external database has information for both.

7.9.4.8. Object attributes data source

An object attributes data source retrieves data from a database and displays it as either object attributes or Smart Point attributes. From the user's perspective, these attributes are just like any other attributes stored in the 3D model, only their values can change dynamically according to data in the external database.

7.9.4.8.1. Creating an object attributes data source

Create an object attributes data source to enable 3D objects to retrieve attribute values from an external database.

Prerequisites

- To create a data source for Smart Points, the required Smart Point Type has been created as described in Creating a Smart Point type.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. On the **Adapters** list, click the database adapter to which to add the data source.
4. In the adapter settings, click **Add data source**, select the data source type **Database Object Attributes Data Source**, and then click **Create data source**.
5. In the **Data Source** section, specify the following settings:
   - **Name** – Enter a name for the data source.
   - **Description** (optional) – Enter a description for the data source.
   - **State** – Specify whether the data source is disabled or enabled (default).
6. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
   - If no groups are selected, only administrators will see the data.
   - Add user groups with **Add**.
   - Remove user groups with the remove button.
   - If the **All Users** group is selected, other group selections will be ignored.

   The **Adapter Configuration** section is read-only.

7. In the **Data Source Configuration** section, specify the following settings:
    - **Provide data for** – Select whether to retrieve the data to objects or Smart Points.
        - **Model Objects** – the retrieved data will be added to objects that have a specific attribute.
        - **Smart Points** – the retrieved data will be added to Smart Points of the specified type.
    - **Parameter Set**– Enter the attributes for which the data source is to provide data. There can be multiple attributes in a parameter set, and multiple parameter sets. To add another parameter set, click **Add Alternative Parameter Set**. This option is visible if you set **Provide data for** to **Model Objects**.
    - **Smart Point type served**– Select the appropriate Smart Point from the list. This option is visible if you set **Provide data for** to **Smart Points**.
    - **Query Builder**– If the selected database type is Microsoft SQL Server, you can select to use Query Builder. Set to **Enabled** to use a simple UI to build the query. If you are using the query builder, select the following:
        - **Database**– Select a database to query data from.
        - **Table**– Select a table to query data from. Select **Join Table** to add inner joins (returns rows with matching values in both tables), left joins (returns all rows from the left table, and only matched rows from the right table), right joins (returns all rows from the right table, and only matched rows from the left table), and full joins (return all rows when there is a match in either table). Select the fields from tables.
        - **Fields**– Specify which fields or columns to select for the query. Leave empty to select all the fields.
        - **Filtering Rules**– Specify filtering rules for the query. Select **Add Condition** to add another condition for the rule. Select **AND** or **OR**, which is then used also for the following conditions.
        - **Number of Rows to Select**– Specify how many rows from the top should be returned from the query. Leave empty to select all rows.
        - **Order Results By**– Specify how the query results are sorted. Select **Ascending** or **Descending**.
    - **SQL Query** – Enter the SQL query to use to retrieve data. If you are using query builder, the SQL query is created based on the selections.

- **Test Key** – There will be multiple fields, if multiple parameter keys are used in the query.

8. Click **Execute Query** to test the configuration.
9. In **Displayed Attributes** you can categorize the attributes, or rename the columns in **Display Name** field.
    - If the type of the attribute is **Value (String)**, you can select it to be output as a hyperlink by changing the type as **Hyperlink** from the drop-down menu in **Type**. The visible hyperlink text is set in **Display Name** field.
        - If set as **Hyperlink**, select how the hyperlink should open:
            - **Always in a new tab** – Each hyperlink opens a new tab (or browser window).
            - **In the data source's own tab** – All hyperlinks of this data source use the same tab.
            - **In the named tab** – All hyperlinks of this data source, as well as of any other hyperlink data source that is configured to use the same tab name, use a named tab. (Specify the tab name to use.)

              To target the current tab, name the tab as "_self".
10. Set **Allow Search Queries**to **Enabled** if the data source can be used in search queries.

    If search queries are enabled, define **Maximum Number of Data Source Queries**, which determines the number of queries the data source can be included in. The field can be left empty for no limit. The default is 250.
11. Click **Save**.

Selecting attributes

When a user working with a project chooses to see the details of an object in the 3D model, Cadmatic eShare looks up the attributes the object has and makes data queries for those attributes. If a data source has any of these attributes in **Parameter Set** configuration, then the data source is queried with the values of the attributes from the model object, and the results are shown to the user.

> Example: The model contains an object with the attribute *Valve Position ID* having a value of 12345. A data source has **Parameter Set** set to *Valve Position ID*. When the user asks for details of the object, the data source will perform its query with a key value of 12345 and returns attributes if the database contains data for this key.

Multiple parameter sets are useful if there are several attributes that identify items of the same general kind. For example, you might want to use the same data source for equipment and valves if the external database has information for both.

Attributes and searching

When users search for objects using ids, Cadmatic eShare will not only look into the internal model but also asks from all Object Attributes data sources if they provide data for this id. In case a data source returns attributes, it is concluded that this id exists in an external system and will be shown in search results. The user is then able to navigate to the object and see data provided by the data source.

Data sources that serve multiple attribute sets will return multiple external references – one for each attribute set. So, if a data source serves for example a set containing Equipment Position Id (nam), another set containing Valve Position Id (vpo), and a third set containing Instrument Position Id (ipo), and the user performs a search for some value (such as P100) that the given data source provides data for, all three attributes will appear as external references in the search results. To avoid this, create more specific data sources that serve data for single attributes. In the example above, create three separate data sources, one for each of Equipment Position Id, Valve Position Id, and Instrument Position Id, with more specific data queries that take into account the type of the searched data.

It is possible to create a data source that provides data for an attribute that is not the id of an object. For example, objects may have an item code that acts as a reference to a warehouse inventory system and can be used to query information from the inventory database. The attribute is considered to be an id of an external object, in this case the material. When users search for ids with an item code, Cadmatic eShare shows a reference to the material as if it was an object.

7.9.4.9. SQL queries

A data source defined in Cadmatic eShare can be seen as a question such as “If I have an object with tag ‘sys’ having value ‘12345’, what is the delivery status of this object?”. An SQL query is a formal way of representing such question. The exact format of the query might depend on database type, but simple SQL queries are usually similar in different databases.

A basic SQL query uses the following structure:

```
SELECT ColumnName1, ColumnName2
FROM TableName
WHERE ObjectId = @Key
```

When this kind of query is executed in Cadmatic eShare, *@Key* is replaced with the value of the object tag that triggered the query. The list of column names defines the columns in the result.

Defining display attributes

Once you have defined the SQL query, enter a value in the **Test object ID** field and click **Execute Query**. This populates the **Displayed Attributes** table, and you can now fill in the Categories and Display Names that are used when showing the resulting attributes to the user. When displaying attributes, they are grouped using the categories. If no display names are defined, the attributes are shown with their column names.

The test id for the query needs to be valid, meaning that it needs to have the correct data type. If you enter a value that does not exist in the database, the Displayed Attributes table will be updated but the Test Results will be empty. If you enter a value that does exist, the Test Results shows the data returned by the query.

7.9.4.9.1. Example SQL queries

SQL queries can be written on a single line, but here they are displayed on multiple lines to enhance readability.

Values from the same table as the key

```
SELECT DeliveryStatus, OrderDate
FROM ErpData
WHERE id = @Key
```

This query returns the columns ‘DeliveryStatus’ and ‘OrderDate’ from the database table ‘ErpData’, from the row where the value of the ‘id’ column is the value of the tag served.

Values from different tables

```
SELECT DeliveryData.DeliveryDate, OrderData.Order
FROM DeliveryData
INNER JOIN OrderData
ON DeliveryData.OrderId = OrderData.Id
WHERE id = @Key
```

This query assumes that you have the tables ‘DeliveryData’ and ‘OrderData’ in your database. ‘DeliveryData’ table has columns ‘id’, ‘OrderId’, and ‘DeliveryDate’. ‘OrderData’ table has columns ‘Order’ and ‘Id’.

The query finds the row in ‘DeliveryData’ in which the ‘id’ column has the value of the tag. It then finds the row from the ‘OrderData’ table where the ‘Id’ column has the same value as the ‘OrderId’ column in the row in ‘DeliveryData’. The displayed values will be selected from the ‘DeliveryDate’ and ‘Order’ columns of the two tables.

Values from different tables with identical column names

```
SELECT DeliveryData.OfficialDate as deliveryDate, OrderData.OfficialDate as orderDate
FROM DeliveryData
INNER JOIN OrderData
ON DeliveryData.OrderId = OrderData.Id
WHERE id = @Key
```

This query assumes that you have the tables ‘DeliveryData’ and ‘OrderData’ in your database. ‘DeliveryData’ table has columns ‘id’, ‘OrderId’, and ‘OfficialDate’. ‘OrderData’ table has columns ‘OfficialDate’ and ‘Id’.

The query finds the row in ‘DeliveryData’ in which the ‘id’ column has the value of the tag. It then finds the row from the ‘OrderData’ table where the column ‘Id’ has the same value as the ‘OrderId’ column in the row in ‘DeliveryData’. The result is that the values from the ‘OfficialDate’ column of both tables will be selected. The Displayed Attributes table of Cadmatic eShare will show these as ‘deliveryDate’ and ‘orderDate’.

Tag calue divided to values of multiple columns

```
SELECT Standard
FROM Pipes
WHERE Area = SUBSTRING(@Key, 1,2) AND Line = SUBSTRING(@Key,6,5)
```

This query compares parts of the value of the tag with two different columns in the database. It assumes that the value of the tag has 10 characters, and that the database has the table ‘Pipes’ with the columns ‘Standard’, ‘Area’, and ‘Line’. The value of the pipeline id is contained in the Area and Line columns.

The query compares the first five characters of the tag value with the ‘Company’ column, and the last five characters of the tag value with the ‘Location’ column. The return value is the value of the ‘Email’ column in such a row.

Please note that this example is for Microsoft SQL Server. Other database types might have different methods for string handling.

7.9.4.9.2. More complex SQL queries

Depending on the database or system that the information comes from, formulating the SQL queries might not always be straightforward. In this section we cover some cases that could be encountered when creating categorization for external data sources. Basically any case where it's possible to write a SELECT clause that produces results with the category and the required Tag values can be used as a categorization data source.

The examples are presented using the SQL syntax of Microsoft SQL Server. In some of the more complex cases, the format of the SQL statements might depend on the database system.

Required data is split over multiple tables

Sometimes the required information cannot be found from a single database table, and the query needs to refer to multiple tables. For example, valve positions might be mapped to internal object identifiers in *IdTable*, while the actual category values are defined in *StatusTable* using the internal IDs instead of the original valve positions. This could be handled with the following query.

> Tags: vpo
>
> Query: *SELECT StatusTable.Status, IdTable.ValvePosition*
> *FROM IdTable, StatusTable*
> *WHERE IdTable.InternalId = StatusTable.InternalId*

Tag value is split into multiple columns

Sometimes database column values might not directly match Tag values. For example, the System (sys) tag in the model can consist of name and ID separated by an underscore (‘Maintenance_001’), but in the database the name and ID might be stored in separate columns. This could be handled by combining the column values in the query, as shown below.

> Tags: sys
>
> Query: *SELECT Status, (SystemName + '_' + SystemId) AS CombinedName*
> *FROM SystemTable*

Now the values of sys are matched against the returned CombinedName column.

Multiple result rows for each object

Sometimes a database table might have multiple entries for a single object, with different values. For example, a table might contain the results of valve safety inspections, including all previous

inspections. For categorization, you would only want to display the results of the latest inspection, to show the current status of an item.

Tags: vpo

Query: *SELECT Status, ValvePosition*
FROM ValveTable AS T
WHERE InspectionDate =
(SELECT MAX(InspectionDate)
FROM ValveTable
WHERE ValvePosition = T.ValvePosition)

Database values are not directly mapped to categories

Sometimes categorization needs to be using value ranges, instead of the individual values of a single database column. For example, you might want to categorize pipelines based on their length, but not have a separate category for every possible length. In this case, the query could specify the categories for level ranges and return that as the first column of the results, as described below.

Tags: pli

Query: *SELECT CASE WHEN PipeLength BETWEEN 0 AND 9 THEN 'Short' WHEN PipeLength BETWEEN 10 AND 19 THEN 'Medium' ELSE 'Long' END AS LengthRange, Pipeline FROM PipeTable*

This query adds the pipelines with length between 0 and 9 to category “Short”, pipelines with length between 10 and 19 to category “Medium”, and the rest into category “Long”.

Combining categories

Category definitions can use the same display name for several categories, to combine them into a single category. This can specified manually or by using a query that returns the same value for several items, in a manner similar to the previous example. For example, you might want to combine the status column values ‘Damaged’ and ‘Disconnected’ into ‘Needs Maintenance’, as described below.

Tags: ipo

Query: *SELECT CASE WHEN Status = ‘Damaged’ THEN 'Needs Maintenance'*
WHEN Status = ‘Disconnected’ THEN 'Needs Maintenance'
ELSE Status END AS CategoryValue, InstrumentPosition
FROM InstrumentTable

7.9.4.9.3. Using multiple queries

When one query is not enough to provide all the required categorization data, you can add multiple queries to a single categorization data source. For instance, you can use this when categorization information is stored in different ways or in different tables.

For example, categorization might be based on the status of equipment, but the database might not have the status as a separate column anywhere, but instead have a table with the list of all the damaged components, another for components that have been ordered but not yet delivered etc. You can solve this case by making a separate query to each of the tables.

Query 1:

Tags: ipo

Query: *SELECT 'Damaged' AS Status, InstrumentPosition FROM DamagedInstrumentsTable*

Query 2:

Tags: ipo

Query: *SELECT 'Ordered' AS Status, InstrumentPosition FROM InstrumentOrdersTable*

This configuration would cause both queries to be performed with instruments found in the DamagedInstrumentsTable to be placed in the Damaged category and instruments from InstrumentOrdersTable to be placed in the Ordered category.

When using multiple queries, it is also possible to have them based on different tags, or even on Compound Tags with a different number of Tags (within a single query, Compound Tags must still consist of the same number of Tags as always). This can be useful when the same categorization needs to be used for different kinds of items, for example for Valves (using Valve Position Tag vpo) and for Spools (using a Combination of Isometric Drawing Number Idn and Spool Number spn). For example:

Query 1:

Tags: vpo

Query: *SELECT Status, ValvePosition FROM ValveStatusTable*

Query 2:

Tags: Idn, spn

Query: *SELECT Status, DrawingNumber, SpoolNumber FROM SpoolStatusTable*

7.9.5. Excel adapter

Excel adapter allows Cadmatic eShare to access a Microsoft Excel or a CSV file, retrieve data rows from a specific worksheet, and add retrieved data to object properties, use it to categorize objects based on visual styles or object hierarchies, or to create a history graph based on the values.

You need to create a separate adapter for every Excel or CSV file that you want Cadmatic eShare to access. One adapter can have multiple data source configurations, meaning that Cadmatic eShare can retrieve different types of data sets from a single Excel or a CSV file.

If the Excel or CSV file contains hyperlinks to web pages or files in a file system, eShare detects the links automatically. The user can open the links from the object's properties pane.

The supported file types are XLS, XLSX, XLSM, and CSV.

7.9.5.1. Creating an Excel adapter

Create an Excel adapter to enable Cadmatic eShare to access a specific Microsoft Excel or a CSV file and read data from a specific worksheet. Note that the adapter also reads data from rows and columns that in Excel are set to be hidden or filtered out.

Prerequisites

- The system administrator has configured the path to the folder in the application settings. If the system administrator has not configured the path, eShare does not have the **Browse** button for selecting the file. However, you can enter the path to the Excel or CSV file manually.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. In the **Adapters** section, click **Add**.
4. In the **Available Adapter Types** section, select **Excel Adapter**, and then click **Create adapter**.
5. In the **Adapter** section, enter a name and description (optional) for the adapter, and specify whether the adapter should be enabled.
6. In the **Adapter Configuration** section, specify the following settings.

- Excel file
  - If eShare has the **Browse** button, click the button to select the file. eShare lists the available folders, Excel, and CSV files. The supported Excel file types are XLS, XLSX, and XLSM.
  - Select the file and click **Update**.
  - To remove the selected file, click **Clear**.
  - If eShare does not have the **Browse** button, enter the location of the Excel or CSV file on the Cadmatic eShare server (local file system path) or on a network disk (UNC path). You can click the **Update** button to check that the path is correct and to retrieve sample data from the file.
  - If the selected Excel file is located outside the folders set in *Cadmatic.SharedFolderRootPath* in application settings in Internet Information Services (IIS) Manager, or given as a relative path, an error specifying the folders is shown.
- **Worksheet name** – Specify the name of the worksheet to use (*Sheet2*, for example), or leave empty to use the first worksheet of the file.
- **First table row number** – Specify the row number from which to start reading data.
- **Number of data rows to skip in the end** – If the data rows to be read are followed by rows that should be ignored, specify the total number of rows to ignore; empty rows are automatically ignored. 0 (default) indicates that all data rows are to be read.
- **Has header row?** – Specify whether the first data row should be used as a header row.
  - **No** – The system uses the default column names from Excel (A, B, C...).
  - **Yes** – The system takes column names from the first data row.

  Regardless of this setting, you can override the default column names with custom names, either in adapter settings or in data source specific settings.
- **Is autoreloading?** – Specify whether the Excel or CSV file should be monitored for changes.
  - **No** – The Excel or CSV file is not actively monitored for changes; an administrator must open the adapter settings and click **Update** to refresh the data.
  - **Yes** – The system periodically checks the Excel or CSV file for changes and automatically retrieves data if changes are detected. You should select this option if the Excel or CSV file connects to an external data source and automatically reloads data, or if you otherwise expect the file to change often.

Note:
This automatic loading of changed data will fail if a user has the Excel or CSV file open. Once the file is closed, the next periodic check should again be able to read data from it.

- **Columns**– Optionally, specify the column and category names to display to users.
  - **Number** – This column displays the sequence number of each data column in Excel or CSV file.
  - **Name** – This column displays the default name of each column, using either default Excel column headers or column headers defined in the file, as specified by the 'Has header row?' setting.
  - **Display Name** – In this column you can provide a custom display name for each imported data column. If not defined, the value displayed in the Name column will be used.
  - **Display Category**– In this column you can provide a custom display category for each imported data column. If not defined, the adapter's name will be used as category name.
- **Extraction Rule for Column Parts** – Select **Add new rule** to create a new rule for extracting parts of the columns in the Excel file with regular expressions. Only existing columns in the Excel file can be used as source columns. Select the column in **Source column** and enter the regular expression in **Pattern** field. Click the **Test** button to test the pattern against the data found in the Excel file. Multiple extraction rules can be configured, but it is not required to use the rules.
- **Define Derived Columns** – Define the columns derived with the extraction rules. Select **Add new definition** to create a new definition for the derived columns. Enter the name of the new column in the **Column Name** field. Define the format of the new column in **Format** field. Possible values for the field are the original columns from the Excel file (displayed with the column number as a prefix), column parts extracted in **Extraction Rules for Column Parts** section, and free text. After configuring all the definitions for the derived columns, select **Update Columns**.

  The new derived columns appear in a separate **Derived Columns** section below the **Original Columns** section.
- **Revisioning Type**– If the Excel or CSV file contains multiple rows that represent different revisions of the same data, set up the revisioning settings to allow only the

latest revision to be imported. First, select the type as either "Major/minor" or "Custom list" and then the other settings as appropriate.

- **Revisioning Column** – Select the column that contains revision numbers.

  If you set Revision Type to "Custom list", specify this setting:

  - **List of Revisions** – Type the possible revision values as a comma-separated list, in ascending order from the first revision to the latest revision. For example, *1,2,3* or *Not Started,Started,Completed*.

  If you set Revision Type to "Major/minor", specify these settings:

  - **Revision has minor part?** – Set this to **Yes** if revision is identified with two numbers or alphanumeric strings that have a specific separator character or string between them.
  - **Is minor part optional?**– Set this to **Yes** if revision does not necessarily have a minor part. For example, if revision can be *1*, *1-A*, or *1-B*.
  - **Revision separator**– If the revision has a minor part, specify the character/string that separates the two parts.
  - **Revision major part type**, **Revision minor part type** – Select whether revision type is specified with a number or an alphanumeric string.
- **Data**– Displays the assigned column names and sample data rows retrieved from the file; click the **Update** button of the Excel or CSV file field to re-import the data, if needed, and check that data is retrieved as expected.

7. Click **Save**.

Related Actions

- To use the data source for assigning values to object attributes, see Creating an Excel object attributes data source.
- To use the data source for creating visual styles or object hierarchies, see Creating an Excel categorization data source.
- To use the data source for creating a history graph, see Creating an Excel historical data source.

7.9.5.2. Creating an Excel object attributes data source

An object attributes data source retrieves data from an Excel or CSV file and displays it as additional object attributes in the 3D model. From the user's perspective, these attributes are just like any

other attributes stored in the 3D model, only their values can change dynamically according to data in the external file.

In the data source settings, you specify which Excel or CSV file column to map to which model attribute; these mappings function as object identifiers, and data is retrieved from the file only if object has all the mapped attributes.

In addition, in the data source settings you can define filters that require specific values to exist in the Excel or CSV file; data is retrieved from the file only if matching rows contain the values specified with filters.

Prerequisites

- The columns in the Excel or CSV file have been named so that you know which attribute they can be mapped to.
- To create a data source for Smart Points, the required Smart Point Type has been created as described in Creating a Smart Point type, and the Smart Point Type has the **External ID** option set to 'Yes'.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. On the **Adapters** list, click the Excel adapter to which to add the data source.
4. In the **Data Sources** section of the adapter settings, click **Add data source**.
5. Select **Excel Object Attributes Data Source**, and click **Create data source**.
6. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
7. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.
    - Remove user groups with the remove button.
    - If the **All Users** group is selected, other group selections will be ignored.

8. The Adapter Configuration section is read-only.
9. In the Data Source Configuration section, specify the following settings:
    - **Provide Data for** – Select whether to create the mapping to model objects or Smart Points.
        - **Model Objects** – If selected, the mapping between an excel file column and a model attribute needs to be specified. Multiple mappings can be specified. All the mappings must match a model object and an Excel or CSV file row to establish a link.
        - **Smart Points** – If selected, the Smart Point Type and the column ID for linking row data to Smart Points need to be specified. The value in the column must match the external ID of the Smart Point to establish a link.
    - **Mappings** – Click **Add Mapping** to map an Excel or CSV file column to an object attribute.
    - **Data Filters** (optional) – Click **Add Filter** to specify that only rows that contain a specific column value will be considered; select a column, and specify the value to look for.

      Example: If column 1 of the Excel file is mapped to the Pipeline attribute, column 2 is mapped to the System attribute, and a data filter specifies that the value of column 4 must be 'PL', an object retrieves attribute data from the Excel only if the object has the attributes Pipeline and System, and if the value in column 4 of the Excel file's matching row is 'PL'.

    - **Columns**
        - You can specify a data source specific Display Name for each column.
        - You can specify a data source specific Display Category for each column.
        - Clear the 'Is Visible?' check box if you do not want a specific column to be displayed as an object attribute in the 3D model viewer. To select or deselect all columns, click the 'Is Visible?' check box in the header row.
    - **Allow Search Queries**– Select **Enabled** if the data source can be used in search queries. If search queries are enabled, define **Maximum Number of Data Source Queries**, which determines the number of queries the data source can be included in. The field can be left empty for no limit. The default is Unlimited.
10. Click **Save**.

Results

When the user selects an object in the 3D view and the object's attributes match the data source settings, the property pane displays additional attributes using data retrieved from the Excel or CSV file specified in the Excel adapter.

7.9.5.3. Creating an Excel categorization data source

Add a categorization data source to an Excel adapter to allow objects in the 3D model to be categorized based on data retrieved from Excel or CSV file. When such categorizations exist, the user can select a suitable category from the visual style or hierarchy drop-down menu of the Model tab, and then view objects that relate to the given category.

In the data source settings, you specify which Excel or CSV file column to map to which model attribute; these mappings function as object identifiers, and data is retrieved from Excel or CSV file only if object has all the mapped attributes. In addition, you can define filters that require specific values to exist in the Excel or CSV file; data is retrieved from the file only if matching rows contain the values specified with filters.

In the categorization settings you can use the values retrieved from the file to display each value as a separate category or you can group values to create fewer, more appropriate categories.

Prerequisites

- The columns in the Excel or CSV file have been named so that you know which attribute they can be mapped to.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. On the Adapters list, click the Excel adapter to which to add the data source.
4. In the Data Sources section of the adapter settings, click **Add data source**.
5. Select **Excel Categorization Data Source**, and click **Create data source**.
6. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).

7. In the Allowed for User Groups section, select the **Is Allowed** check box of the user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - If the All Users group is selected, other group selections will be ignored.
8. The Adapter Configuration section is read-only, it displays the adapter configuration.

    Next, specify the Data Source Configuration settings.

9. **Caching** – This setting defines if data source categorization is always fetched from the data source or if a cached categorization is used.
    - **Disabled** – (default) – The data source categorization is always fetched from the data source.
    - **Enabled** – The data source categorization is fetched from the data source when eShare is started or when a data source is configured. After that caching is renewed in intervals defined in **Caching Interval** field. The value is given in minutes.
10. **Visual Style** defines whether the categories created by this data source configuration are to be displayed in the visual style drop-down menu.
    - **State** – When this setting is enabled (default), the user can select the category from the visual style drop-down menu, and the 3D view highlights objects with value-specific colors.
    - **Conflict resolution** – This setting specifies where an object should appear if it matches several categories: in the first category, the last category, or the special Multiple Categories category.
11. **Hierarchy** defines whether the categories created by this data source configuration are to be displayed in the hierarchy drop-down menu.
    - **State** – When this setting is enabled (default), the user can select the category from the hierarchy drop-down menu, and the Models tree lists objects in attribute value specific nodes.
    - **Conflict resolution** – This setting specifies where an object should appear if it matches several categories: in all categories, the first category, the last category, or in the special Multiple Categories category.
12. **Queries** define the data that must be found in the Excel or CSV file for categories to be created. Click **Add query** and then specify the following settings.
    - **Mappings** – Click **Add Mapping** to map an Excel column to an object attribute (their values must match for the link to be established).

- **Data Filters** – Click **Add Filter** to specify that only rows that contain a specific column value will be considered; select a column, and specify the value to look for.
- **Value column** – Select the Excel or CSV column from which to take values for creating the categorization. The name of the column will be the default name of the category.

13. **Categories** defines how the data is categorized. The values in the Value column must match exactly the values found in Excel or CSV file, unless you use value ranges or regular expressions as described below.

    The **Case sensitivity** setting specifies if the value column in the defined categories is expected to be case-sensitive when matching the values to the found values. Case sensitivity does not affect fields that use ranges or regexes (they are always case sensitive). For categorizations created in earlier releases, case sensitivity is disabled by default.

    The **Show to user** setting specifies what categories are displayed to the user in the visual style and hierarchy drop-down menus.

    - If set to 'Listed categories', the user will only see the categories that are defined in this view.
    - If set to 'All categories', the system will create a category for each value that is not defined in this view, and the user will see both manually configured and automatically generated categories. Because these dynamic categories are created on-the-fly and not saved, the color that is assigned to them in visual style might change if the values change.

    First, you can click **Refresh** if you want to see the categories that are created based on the queries you just defined. Then, edit the categorization as needed.

    In the simplest scenario, the values retrieved from Excel or CSV file are used as categories as is. Optionally, you can enter a suitable Display Value for each category, define the category color (only relevant if the categories are used as visual styles), and specify the relative order of the categories.

    - You can specify the same Display Value to multiple Excel or CSV values, to include the values in the same category.
    - *Multiple Categories* settings will be used for objects that fit in multiple categories.
    - *Uncategorized* settings will be used for objects that do not belong to any category.

    Or, you can use the Value field to manually define the categorization as follows.

    - Enter a specific value.
      If the value is not present in the Excel or CSV file, clicking **Remove Unused** or **Refresh**

will remove the category.

- Enter the equals sign (=) and a specific value.
  Clicking **Remove Unused** or **Refresh**will keep the category, even if the value is not present in the Excel or CSV file.
- Define a range of values that will be included in the same category.
  Define the range using the format *from ... to*. The first value (*from*) is included in the range, but the second value (*to*) terminates the range and is not included in it. Either value can also be omitted to leave one end of the range open. (Spaces around the three dots will be ignored.) Examples:
  a ... m matches the letters from a to n
  0 ... 3 matches values such as 0, 1.5, and 2.999
  ... 2017-10-15 matches any date before 2017-10-15
  2017-10-15 ... matches 2017-10-15 and any date after it
- Define a regular expression that matches the required values. Start and end the regular expression with the slash character (/). Examples:
  /[ABC]\d{1,4}/
  /2017-10-\d+/

14. Click **Save**.

Results

The user can select data source specific visual styles and/or object hierarchies from the **Model** tab of the 3D model viewer.

7.9.5.4. Creating an Excel historical data source

A historical data source retrieves history data for objects or groups from an Excel or CSV file and displays it as a history graph in the properties pane of the object or group. From the user's perspective, these attributes are just like any other attributes stored in the 3D model, only their values can change dynamically according to data in the external file.

In the data source settings, you specify which Excel or CSV file column to map to which model attribute; these mappings function as object identifiers, and data is retrieved from the file only if object has all the mapped attributes.

In addition, in the data source settings you can define filters that require specific values to exist in the Excel or CSV file; data is retrieved from the file only if matching rows contain the values specified with filters.

Prerequisites

- The columns in the Excel or CSV file have been named so that you know which attribute they can be mapped to.
- To create a data source for Smart Points, the required Smart Point Type has been created as described in Creating a Smart Point type, and the Smart Point Type has the **External ID** option set to 'Yes'.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. On the Adapters list, click the Excel adapter to which to add the data source.
4. In the Data Sources section of the adapter settings, click **Add data source**.
5. Select **Excel Historical Data Source**, and click **Create data source**.
6. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
7. In the Allowed for User Groups section, select the **Is Allowed** check box of the user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - If the All Users group is selected, other group selections will be ignored.
8. The Adapter Configuration section is read-only, it displays the adapter configuration.

    Next, specify the Data Source Configuration settings.

9. **Provide data for** – Select whether to create the mapping to model objects or Smart Points.
    - **Model Objects** – If selected, the mapping between an excel file column and a model attribute needs to be specified. Multiple mappings can be specified. All the mappings must match a model object and an Excel or CSV file row to establish a link.
    - **Smart Points** – If selected, the Smart Point Type and the column ID for linking row data to Smart Points need to be specified. The value in the column must match the external ID of the Smart Point to establish a link.
10. **Mappings** – Click **Add Mapping** to map an Excel or CSV file column to an object attribute.
11. **Data Filters** (optional) – Click **Add Filter** to specify that only rows that contain a specific column value will be considered; select a column, and specify the value to look for.
    Example: If column 1 of the Excel file is mapped to the Pipeline attribute, column 2 is mapped to the System attribute, and a data filter specifies that the value of column 4 must be 'PL', an object retrieves attribute data from the Excel only if the object has the attributes Pipeline and System, and if the value in column 4 of the Excel file's matching row is 'PL'.
12. **Time Column** – Select which column's values are used as the time value for the graph, and whether the value is represented in **Local time** or **UTC**.
13. **Data Column** – Select which column's values to use as the data value for the graph. The data can be numeric or enumerated.
14. **Display Name** (optional) – You can specify a custom display name for the graph.
15. **Display Category** (optional) – You can specify a custom display category for the graph.
16. **Unit** (optional) – You can specify a data unit for the graph.
17. **Is Enum** – If set to **Yes**, and data in the configured data column is enumerated type, the **Detected Enum Values** table will list the distinct values found in the data column. The fetching and detecting of these values is not done automatically and the **Refresh** button has to be pressed every time a new type of enum values appear or older ones disappear from the data column.
    The data in the Detected Enum Values table can be modified by dragging the values to the desired level. The enum values will be presented in the same order in the resulting graph's y-axis as they are in the table. Meaning that the top most value in the table will be the highest value in the graph and the last value in the table will be the lowest value in the graph.
18. Click **Save**.

Results

The user can view a graph of history data in the properties pane for the object or group in the **Model** tab of the 3D model viewer.

7.9.6. File system document adapter

Cadmatic eShare supports a couple of ways of accessing documents stored in a file system. You can create a database adapter with a document data source if the document metadata and folder hierarchy are defined in a database, as described in Document data source, but a much easier method is to use a file system document adapter.

You can enable Cadmatic eShare to access documents in a file system simply by defining the document base path in the adapter's data source configuration. To enable the system to also create links from 3D objects to relevant documents in that location, you must define suitable path detection patterns.

Note that if you want the **Documents** view to provide additional information about the documents in the tree, the file system document data source is limited to metadata provided by the file system, whereas a database document data source can use any attributes that you can retrieve by querying the document database.

7.9.6.1. Creating a file system document adapter

Create a file system document adapter, and then add a data source configuration to it, to enable Cadmatic eShare to connect to a file system and access documents.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. In the **Adapters** section, click **Add**.
4. In the **Available Adapter Types** section, select **File System Document Adapter**, and then click **Create adapter**.
5. In the **Adapter** section, enter a name and description (optional) for the adapter, and specify whether the adapter should be enabled.
6. Click **Save**.

Related Actions

Add a data source to the adapter as described in Creating a file system document data source.

7.9.6.2. Creating a file system document data source

Add a data source to a file system document adapter to enable the adapter to provide access to document files and to enable objects in the 3D model to be automatically linked to documents.

Prerequisites

- You know the base path to project documents stored in the file system.
- To enable indexing, a suitable document handler must be defined as described in Document handlers. For example, the following settings allow all documents under the base path to be indexed:
  - Detection of Document Type

    Identifier Name: Path

    Pattern to Match: .*
  - Document Processing Rules

    Detection of Model Objects: Automatically
- To define path detection patterns, you need to know what kind of information is stored in the file path or file name of documents, and how that information can be mapped to the attribute values of objects in the 3D model.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The **Adapters and Data Sources** configuration view opens.
3. On the **Adapters** list, click the file system document adapter to which to add a data source.
4. In the **Data Sources** section of the adapter settings, click **Add data source**.
5. In the **Available Data Source Types** section, **File System Document Data Source** is already selected—click **Create data source**.
6. In the **Data Source** section, specify the following settings:
   - **Name** – Enter a name for the data source. This name will be shown in the Documents view of the project as the name of the top-level document folder, but it will not be visible in the 3D model viewer.

- **Description** (optional) – Enter a description for the data source.
- **State** – Specify whether the data source is disabled or enabled (default).

7. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
   - If no groups are selected, only administrators will see the data.
   - Add user groups with **Add**.
   - Remove user groups with the remove button.
   - If the **All Users** group is selected, other group selections will be ignored.

   The **Adapter Configuration** section is not relevant for this data source type.

8. In the **Data Source Configuration** section, specify the following settings.
   - **Indexing** — Select if document indexing is enabled. If you enable it, the system can search for object links in documents, store those links in the project database, and use this document index to create links from 3D objects to relevant documents. The documents must be found in the subfolders of the document base path. See Adapters and indexing.

     Note: If you save the data source configuration with indexing, all data sources of the same adapter with indexing enabled, will be indexed again.

     If **Enabled**, specify the following settings:
     - **Indexing Speed**– Define the duration of indexing.

       If you select **Duration**, specify the target duration of each indexing round in **Target Indexing Duration** field (in minutes). The set duration takes effect only after the first full round of indexing has been completed. The default is 120.

       If you select **As Fast As Possible**, the target of each indexing round is to be completed in the shortest time as possible.
     - **Indexing Mode**– Indexing is done every time the data source configuration is saved. Indexing can also be triggered manually from the Project Administration's General view.

       If you select **Once**, indexing is done only after saving.

       If you select **Daily**, indexing is done once every day.

       If you select **Once per Days of Week**, indexing is done once on the selected days.

If you select **Given Times in Days of Week**, indexing starts on given times of the day on the selected days. The times are given in the server's local time.

If you select **Days of Month**, indexing is done once on the selected days of the month.

If you select **All the Time**, indexing is done continuously. Warning: This can consume a lot of resources if As Fast as Possible is selected as indexing speed.

- **Show documents inside ZIP files** — If enabled, also documents that are inside .zip files can be read from the file system. If a .zip file contains documents whose file type is enabled in the File Types setting, the adapter takes the first document from the file. If the .zip contains files of different types, the document is selected using this preference order: .dgn, .dwg, .dxf, .pdf
- **Load Subfolders On Demand** — If enabled, the entire document tree is not loaded at once, but the contents are loaded and visible following the user's examination. The default is **Disabled**.
- **Document Base Path** — This mandatory setting specifies the base location under which documents are stored in the file system. Any subfolders that exist under this top-level folder are displayed as a tree in the Documents view.
- **File Types** — Select which file types to display from the data source: .pdf, .dwg, .dxf, .dgn, Images
- **Additional File Types to Show** — If the document data source adapter should support other file types, list them here. Separate the file types with a comma.
- **Path Detection Patterns** — A path detection pattern allows the system to use .NET regular expressions to match attribute values of 3D objects to document paths in the file system, to automatically link objects and groups to documents. The matching is relative to document base path, and it looks for matches in folder names and file names. Click **Add pattern**, specify the pattern to use, and then click **Validate** to test that the pattern functions as expected.

  You can use object attribute names in the pattern to indicate that that part of the path should match the value of the specified object attribute.

  You can use *Project Name* and *ProjectDatabase Name* as variables in the pattern. These will be replaced with the name of the project and the name of the project database, respectively.

  To add a text string to the pattern, type the text and press Enter.
  Free-form text is displayed with a black background.

To add an attribute to the pattern, start typing the name and then select the appropriate attribute from the drop-down menu.
Attributes are displayed with a blue background.

Examples:

- \System\Position Id\.*



This setting finds documents that are located in any subfolders under a path where the name of the first folder is to match the System of the object, and the name of the second folder is to match the Position Id of the object. If both values match, the object displays links to the documents in this location.

- .*\IsoPipeline.*



This setting finds documents that are located in any level of the folder structure, if the document's filename starts with 'Iso', then contains the value of the Pipeline attribute, and the rest of the filename consists of any string.

9. Click **Save**.
10. If you defined a path detection pattern and used attributes in it, you can check that in the saved configuration any attributes are displayed in curly brackets: *{System}**{Position Id}*\.*.

Results

In the **Documents Browser** of the project, you can browse the folder structure specified in **Document Base Path** and open documents that it contains.

If indexing is enabled, simple search can find indexed documents if the search term is like the value of the link.

When a user examines an object in the 3D view, if the object has properties that can be matched to a document index or path detection patterns, the related documents are listed in the details pane.

7.9.7. Hyperlink adapter

Hyperlink adapter allows an object in the 3D model to be linked to a web address, usually to provide some additional information about that object to the user. Depending on the data source configuration, a hyperlink adapter can function in the following ways:

- When the user selects to examine an object that contains a specific attribute, a hyperlink is displayed in the object details pane.
- When the user adds a Smart Point and selects a Smart Point Type that uses external IDs and is connected to a hyperlink adapter, a hyperlink is displayed in the Smart Point details pane.

The target of the hyperlink can be a static web address, but it can also be a dynamic link to a web-based system that allows item IDs to be passed as part of the URL, so that clicking the link takes the user directly to an object-specific web page. For example, an online parts catalog might accept URLs in the following format:

http://host.com/partscatalog?action=showitem&articlenumber=12345

If this URL format is specified in a hyperlink adapter and the objects in the 3D model have corresponding article numbers stored in a specific attribute, users can click an automatically generated link to open the relevant page of the online parts catalog.

A hyperlink adapter can also include commands in the URL and pass them to the target system, if the target interface allows it.

Usually you do not need to create more than one hyperlink adapter per project, because one hyperlink adapter can contain multiple hyperlink data source configurations each of which defines one specific hyperlink format.

If a hyperlink is valid but does not work, it is possible that Cadmatic eShare considers the protocol to be unsafe. You can allow custom protocols in the AppSettings view of the Internet Information Services (IIS) Manager application by adding a semicolon-separated list of such protocols to the value of the *Cadmatic.ProtocolWhitelist* setting.

7.9.7.1. Creating a hyperlink adapter

Create a hyperlink adapter to enable Cadmatic eShare to create links to web addresses that contain information about objects in the 3D model.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. In the Adapters section, click **Add**.

4. In the Available Adapter Types section, select **Hyperlink Adapter**, and then click **Create adapter**.
5. In the Adapter section, enter a name and description (optional) for the adapter, and specify whether the adapter should be enabled.
6. Click **Save**.

Related Actions

Add a data source to the adapter as described in Creating a hyperlink adapter data source.

7.9.7.2. Creating a hyperlink adapter data source

Add a data source to a hyperlink adapter to enable the adapter to open hyperlinks of a specific format, and to specify whether the hyperlink should be attached to objects or Smart Points.

Prerequisites

- To create a data source for Smart Points, the required Smart Point Type has been created as described in Creating a Smart Point type, and the Smart Point Type has the **External ID** option set to 'Yes'.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. On the Adapters list, click the hyperlink adapter to which to add the data source.
4. In the Data Sources section of the adapter settings, click **Add data source**.
5. In the Available Data Source Types section, **Hyperlink data source** is already selected—click **Create data source**.
6. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
7. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.

- Remove user groups with the remove button.
- If the **All Users** group is selected, other group selections will be ignored.

8. The Adapter Configuration section is read-only.
9. In the Data Source Configuration section, specify the following settings:
   - **Provide data for** – Select whether to add the hyperlink to objects or Smart Points.
     - **Model Objects** – The hyperlink will automatically be included in any objects that have a specific attribute. It also allows you to create dynamic hyperlinks with embedded attribute values.
     - **Smart Points** – The hyperlink will only be included in Smart Points of a specific type.
   - **Smart Point type served** – Select the appropriate Smart Point from the list. (This option is visible if you set **Provide data for** to **Smart Points**.)
   - **Display Category** – Enter the category text, such as “Web Links”, that will be displayed in the details pane of the 3D model viewer. You can also use the same display category in multiple hyperlink data source configurations, for example to display all hyperlinks under the same heading.
   - **Open** – Select how the hyperlinks of this data source should open.
     - **Always in a new tab** – Each hyperlink opens a new tab (or browser window).
     - **In the data source's own tab** – All hyperlinks of this data source use the same tab.

       Note: This option only has effect when the link points to the same origin as eShare server. External links always open in a new tab.
       In eShare App, all links open in a new window regardless of this setting.

     - **In the named tab** – All hyperlinks of this data source, as well as of any other hyperlink data source that is configured to use the same tab name, use a named tab. (Specify the tab name to use.)
       To target the current tab, name the tab as "_self".

       Note: This option only has effect when the link points to the same origin as eShare server. External links always open in a new tab.
       In eShare App, all links open in a new window regardless of this setting.

   - **Address Templates** – Create address templates for the links. You can add multiple address templates, click **Add template** or **Copy** button to add another template. To

remove a template, click the **Delete** button .

- **Link Text** – Enter the link text that will be displayed in the details pane. You cannot use variables in the link text.
- **Target URL** – Specify the URL of the link. Click the field and enter the URL. Click Enter key to add the input as normal text. To add an attribute, begin typing it and select it from the list of available attributes.
- **Additional Required Attributes** (optional) – Attributes that the object must have for the link to be shown, but the attributes are not included in the Target URL. Select the attributes from the drop-down menu. There can be multiple attributes.

- You can use the **Test Link ID** field to test the configuration. Enter any attribute value (it does not need to exist in the model), click **Create Test Links**, and then click the blue link to open it in a web browser.
- **Allow Search Queries** – Select **Enabled** if the data source can be used in search queries. If export to search queries is enabled, define **Maximum Number of Data Source Queries**, which determines the number of queries the data source can be included in. The field can be left empty for no limit. The default is 1000.

10. Click **Save**.

7.9.7.3. Hyperlink adapter example: integration with Google

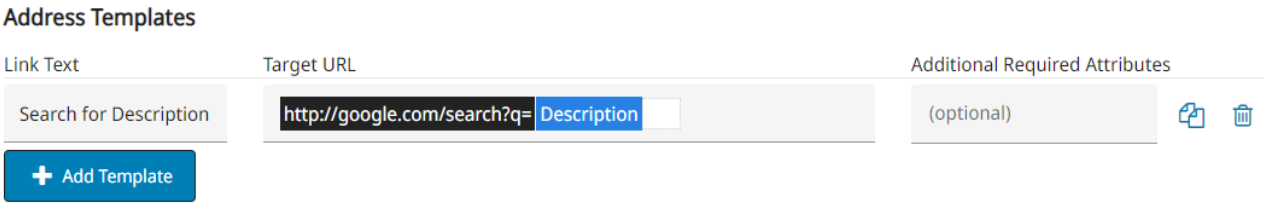
The Google search engine stores search parameters in URLs, which makes it possible to create search URLs manually. For example, this link performs a Google search for the word 'test':

http://www.google.com/search?q=test

You can use this interface to integrate Cadmatic eShare with Google. For example, you can define a link that performs a Google search based on an object's description in the 3D model. In models created with Cadmatic design applications you can refer to the object description with the attribute 'Description'.

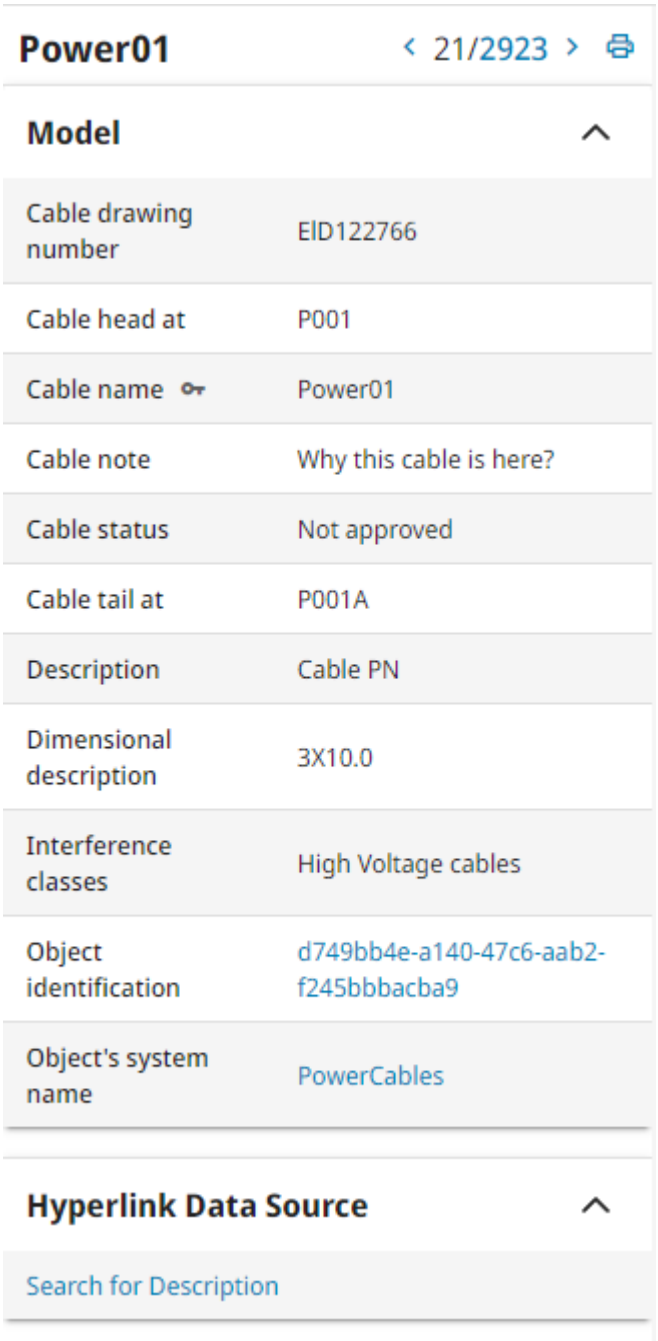
First, you need to create a hyperlink adapter and a hyperlink data source, and make both active. Then, configure the data source as follows:

Provide data for: Model Objects
Display Category: Web Search
Link Text: Search for Description
Address Template: http://google.com/search?q=Description



As a result, the details pane of every object that has an attribute 'Description' displays a 'Search for Description' link, and if the user clicks that link the default web browser of the system opens and searches the web for the value of the Description attribute.

In the example below, clicking 'Search for Description' would search the web for 'Cable PN'.



7.9.8. M-Files adapter

M-Files adapter enables Cadmatic eShare to connect to M-Files document management system, in order to read the folder structure and open PDF documents stored in the folders. If the adapter should support other file types, you can specify them according to the project's needs. Note that

eShare cannot index these other file types, and the user needs to download the files to be able to view them in an external viewer.

7.9.8.1. Creating an M-Files adapter

Create an M-Files adapter to enable Cadmatic eShare to connect to M-Files document management system and access project documents stored in that system.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. In the Adapters section, click **Add**.
4. In the Available Adapter Types section, select **M-Files Adapter**, and then click **Create adapter**.
5. In the Adapter section, enter a name and description (optional) for the adapter, and specify whether the adapter should be enabled.
6. In the Adapter Configuration section, specify the following settings:
   - **M-Files URL** – Public URL of m-files web. This is the same URL that can be accessed using a browser.
   - **Auth Type** – Select **Username/Password** for authentication type. Enter M-Files username and password in the specified fields.
7. Click Refresh button  next to **Vault** field to refresh the available vaults, ie. collections of files in M-Files.
8. Select the used vault from the **Vault** drop-down menu.
   Is Authorized? prompt will show **Yes** if authorized, and **No** if not authorized.
9. After successful authorization, click **Save**.

Related Actions

Add a document data source to the adapter as described in Creating an M-Files document data source.

7.9.8.2. Creating an M-Files document data source

Add a document data source to an M-Files adapter when you want to retrieve documents and their metadata from the external system.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. On the Adapters list, click the M-Files adapter to which to add the data source.
4. In the **Data Sources** section of the adapter settings, click **Add data source**.
5. Select **M-Files Document Data Source** from the list.
6. Click **Create Data Source**.
7. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
8. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.
    - Remove user groups with the remove button.
    - If the **All Users** group is selected, other group selections will be ignored.
9. The Adapter Configuration section is read-only.
10. In the Data Source Configuration section, specify the following settings:
    - **File Type Options**
        - **File Types** – Select the file types that the document data source adapter should support.
        - **Additional File Types to Show**– If the document data source adapter should support other file types, list them here. Separate the file types with a comma.
    - **Indexing Options**
        - **Indexing** – Select if the document indexing is enabled. If you enable indexing, eShare is able to create links between model objects and documents.

Note: If you save the data source configuration with indexing, all data sources of the same adapter with indexing enabled, will be indexed again.

If **Enabled**, specify the following settings:

- **Indexing Speed**– Define the duration of indexing.

  If you select **Duration**, specify the target duration of each indexing round in **Target Indexing Duration** field (in minutes). The set duration takes effect only after the first full round of indexing has been completed. The default is 120.

  If you select **As Fast As Possible**, the target of each indexing round is to be completed in the shortest time as possible.

- **Indexing Mode**– Indexing is done every time the data source configuration is saved. Indexing can also be triggered manually from the Project Administration's General view.

  If you select **Once**, indexing is done only after saving.

  If you select **Daily**, indexing is done once every day.

  If you select **Once per Days of Week**, indexing is done once on the selected days.

  If you select **Given Times in Days of Week**, indexing starts on given times of the day on the selected days. The times are given in the server's local time.

  If you select **Days of Month**, indexing is done once on the selected days of the month.

  If you select **All the Time**, indexing is done continuously. Warning: This can consume a lot of resources if As Fast as Possible is selected as indexing speed.

- Caching Options
  - **Caching** – Select if caching is enabled.

    If **Enabled**, specify the following settings:

    - **Folder Tree Caching** – Define if the folder tree is cached or retrieved from the server.

      If you select **Disabled**, eShare fetches the folder tree from the server every time the user logs in or refreshes the browser.

If you select **Enabled**, eShare saves the folder tree in the cache and updates it only periodically. The user sees the cached folder tree. Caching is renewed in intervals defined in **Folder Cache Time** field. The value is given in minutes.

- **Cache in Background** – Define if the folder tree is cached in the background.

- **Hierarchy Levels** – Creates a document folder hierarchy using metadata.
  - Select **Add Level** to create new hierarchy level.
  - In the drop-down list, select the name of the metadata to create a level from.
  - Refresh the list of metadata from M-Files using the refresh button .
  - **Hide if empty** – If enabled, empty hierarchy categories are hidden.
  - **Display Prefix** – Specify a prefix added to the hierarchy names.

  Levels can be deleted using the delete button. New levels can be added using **Add Level**. Each new level is added inside the previous level. Levels can also be organized by dragging.
- **Document Filters** – Filters documents by metadata and shows only documents matching the filter.
  - Select **Add Filter** to create new document filter.
  - In the drop-down menu, select the name of the metadata to filter out.
  - Refresh the list of metadata from M-files using the refresh button .
  - Select **Is** or **Is Not** in the drop-down menu.
  - Select value of metadata in the drop-down menu. Only works when the metadata has a limited number of values or is true/false.

  Filters can be deleted using the delete button. New filters can be added using **Add Filter**.

  Filters are processed as "and", which means that all documents must match all filters to be shown. However, if you have many filters of the same metadata, M-Files treats this as an OR operator.

11. Click **Save**.

7.9.9. Neo4j adapter

Neo4j adapter allows Cadmatic eShare to access a Neo4j graph database, retrieve data from the database, and add the retrieved data to object attributes or Smart Points.

7.9.9.1. Creating a Neo4j adapter

Create a Neo4j adapter to enable Cadmatic eShare to connect to a Neo4j server using the Bolt protocol.

Prerequisites

- Address, username, and password to the required Neo4j server.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. In the Adapters section, click **Add**.
4. In the Available Adapter Types section, select **Neo4J Adapter**, and then click **Create adapter**.
5. In the Adapter section, specify the following settings:
    - **Name** – Enter a name for the adapter.
    - **Description** (optional) – Enter a description for the adapter.
    - **State** – Specify whether the adapter is disabled or enabled (default).
6. In the Adapter Configuration section, specify the following settings:
    - **Neo4J server address** – Specify the Neo4j server address in this format: *bolt://servername:port*
    - **Neo4J server username** – Specify the username for the Neo4j server.
    - **Neo4J server password** – Specify the password for the Neo4j server.
7. Click **Test** to test the connection.
8. Click **Save**.

7.9.9.2. Creating an object attributes data source for Neo4j adapter

Create an object attributes data source to allow a Neo4j adapter to use a Cypher query to retrieve data from a graph database and add the data to object attributes or Smart Points.

Prerequisites

- To retrieve data to object attributes, you know which tags to use. See Using attributes and tags in adapters.
- To retrieve data to Smart Points, the Smart Point Type to use has **External ID** set to "Yes". See Creating a Smart Point type.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. On the Adapters list, click the Neo4j adapter to which to add the data source.
4. In the Data Sources section of the adapter settings, click **Add data source**.
5. Select **Object Attributes Data Source**, and click **Create data source**.
6. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
7. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.
    - Remove user groups with the remove button.
    - If the **All Users** group is selected, other group selections will be ignored.
8. The Adapter Configuration section is read-only.
9. In the Data Source Configuration section, specify the following settings:
    - **Provide data for** – Select whether the retrieved data is for model objects or Smart Points.

      If you selected "Model Objects", then in **Serving Tags** specify the tags to which to add the data, each tag on a separate row.

      If you selected "Smart Points", then in **Smart Point type served** select the Smart Point Type to which to add the data.
    - **Cypher Query** – Specify the Cypher query to use for retrieving data.

- **Test object ID** – To test the query, enter an object ID that is found in the database and in the 3D model in eShare, and then click **Execute Query**.
- **Displayed Attributes** – Displays the attributes retrieved by the query.
- **Test Results** – Displays test results from the query.
- **Allow Search Queries**– Select **Enabled** if the data source can be used in search queries. If search queries are enabled, define **Maximum Number of Data Source Queries**, which determines the number of queries the data source can be included in. The field can be left empty for no limit. The default is 250.

10. Click **Save**.

7.9.10. PI adapter

PI adapter is an adapter type that allows model objects, and Smart Points that users add to 3D models, to retrieve data from a PI database, using a PI OLEDB connection and an SQL query. The retrieved information is shown to the user in the details pane of the 3D model viewer, and it is constantly refreshed as the adapter polls the database for changes.

7.9.10.1. Creating a PI adapter

Create a PI Adapter to enable Cadmatic eShare to connect to a PI database.

Prerequisites

- The PI OLEDB provider has been installed on the Cadmatic eShare server computer.
- To use Windows Integrated Security authentication, the user account under which the Cadmatic eShare server runs must be defined as a trusted user in PI System Management Tools.
- You have the connection string that Cadmatic eShare needs for connecting to the database. Example: *Provider=PIOLEDB; Data Source=MyServer-01; Integrated Security=SSPI;*

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. In the Adapters section, click **Add**.
4. In the Available Adapter Types section, select **PI Adapter**, and then click **Create adapter**.

5. In the Adapter section, enter a name and description (optional) for the adapter, and specify whether the adapter should be enabled.
6. In the Adapter Configuration section, specify the following:
    - **Connection String** – Specify the connection string to use for connecting to the PI database, and then click **Test** to verify that connection can be established.
    - **Impersonation** – Select if impersonation is enabled or disabled.
7. Click **Save**.

Related Actions

- Add a data source to the adapter as described in Creating a PI adapter data source.

7.9.10.2. Creating a PI adapter data source

Add a data source to a PI adapter to enable the adapter to retrieve data from the PI database. You can also use the data source configuration to test that the system can retrieve data for a specific point ID.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. On the Adapters list, click the PI adapter to which to add a data source.
4. In the Data Sources section of the adapter settings, click **Add data source**.
5. In the Available Data Source Types section, **PI Data Source** is already selected—click **Create data source**.
6. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
7. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.

- Remove user groups with the remove button.
- If the **All Users** group is selected, other group selections will be ignored.

8. The Adapter Configuration section is read-only.
9. In the Data Source Configuration section, specify the following:
    - **Provide data for** – Select whether to retrieve the data to objects or Smart Points.
        - **Model Objects** – the retrieved data will be added to objects that have a specific attribute.
        - **Smart Points** – the retrieved data will be added to Smart Points of the specified type.
    - **Refresh Automatically** – Specifies if visible data should be refreshed automatically in the attribute pane. **Enabled** by default. Set the **Refresh Interval** in seconds. Default is 5 seconds.
    - **Data Time Zone** – Set the used time zone as **UTC** or **Local**.
10. Test the connection by entering a point ID that you know to exist in the database in **Test value**, and click **Execute Query**.
11. Click **Save**.

Related Actions

- Create a new Smart Point Type for PI adapter to allow users to add Smart Points that retrieve PI data. See Creating a Smart Point type.

7.9.11. REST API adapter

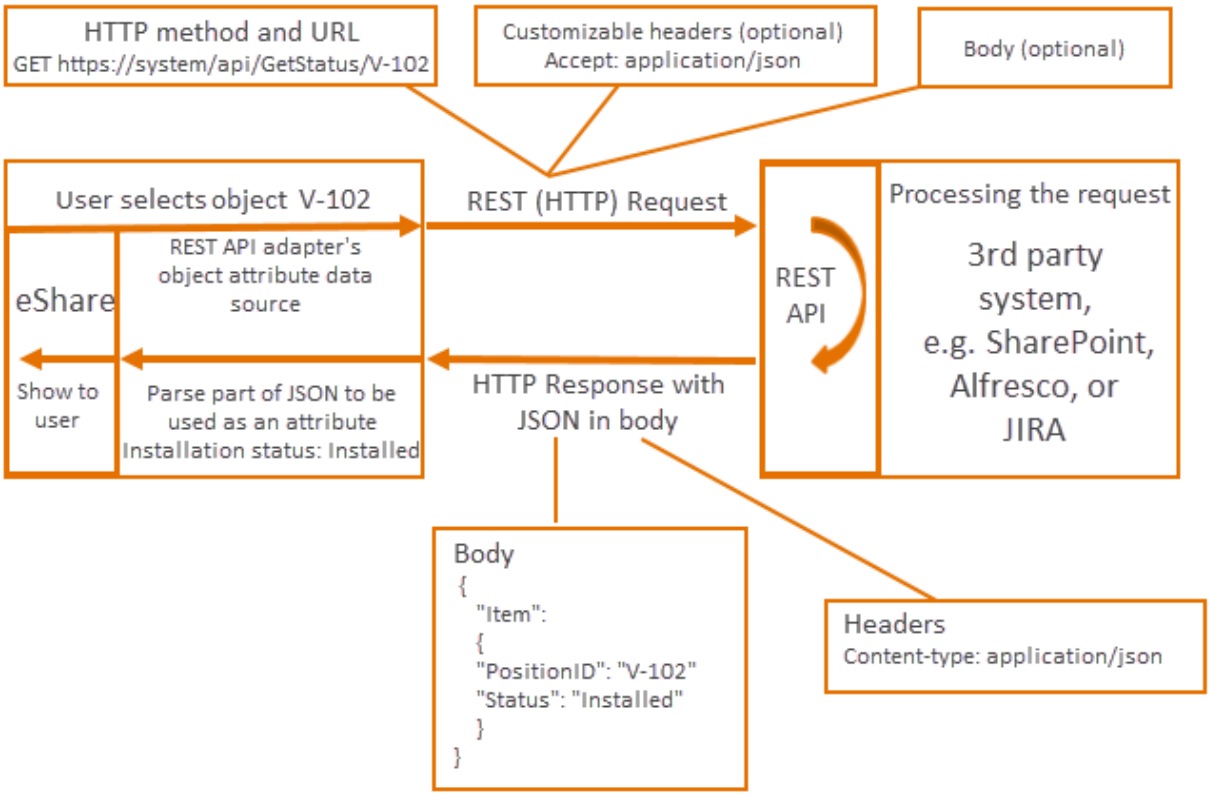
A REST API adapter provides access to services that provide a REST API. After you have successfully configured the REST API adapter, eShare displays the document tree, document list, and document metadata provided by the target service.

The REST API adapter gets information from REST APIs that provide JSON data, and supports different authentication schemes.

Configuring a REST API adapter requires expert-level IT skills, programming experience, and solid experience in eShare administration. To be able to configure a REST API adapter, you need to be familiar with the REST API concept, the principles how REST APIs work, and JSON. You also need to understand the service for which you are configuring the adapter, and be familiar with configuring data sources in eShare.

The REST API adapter works as follows:

1. The user selects an object that is linked to an external service that supports the REST API.
2. eShare sends the request to the external service. The request contains the HTTP method and URL, and if required, customized headers and a body.
3. The external service processes the request.
4. The external service sends eShare an HTTP response. The response contains a header and a body.
5. eShare parses the response.
6. eShare shows the response to the user.



eShare's REST API adapter allows you to test the configuration while you create it.

To configure a REST API adapter:

1. Create the REST API adapter.
2. Add the data source. A REST API adapter must have one or more data sources. The data source type depends on the needs of the project.
3. Test the configuration.

Prerequisites

Familiarize yourself with the target system and its REST API documentation. When you configure the REST API adapter, you need to enter information from the target system. This information is system-specific and project-specific, therefore this documentation cannot provide specific guidelines.

7.9.11.1. Creating a REST API adapter

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. Click **Add**.
4. In the Available Adapter Types section, select **REST API Adapter**, and then click **Create adapter**.
5. In the Adapter section, enter a name and description (optional) for the adapter, and specify whether the adapter should be enabled.
6. In the Adapter Configuration section, specify the following settings:
    - **REST API base URL** – Enter the URL of the system to which the REST API adapter should connect.
    - **Authentication method** – Define the authentication method for accessing the service.
        - **None** – The user can access the service without providing any credentials.
        - **Basic** – Define a custom username and password. You can define only one set of credentials, that is, the credentials are not user-specific but all users use the same username and password.
        - **Windows** – The user can access the service with his or her Windows credentials. If you use Windows authentication, define also if client impersonation is enabled or disabled. If you enable impersonation, the credentials are user-specific. If you disable impersonation, the system will use the Internet Information Services (IIS) identity. Note that not all systems support client impersonation.
        - **Windows (custom)** – Define a custom username and password that work in the domain. You can define only one set of credentials, that is, the credentials are not user-specific but all users use the same username and password.

- **Use Authentication Request** – Authentication requests are needed for session-based services. If set to **Enabled**, select **Auth Type**.
    - If you select **Generic**, the request to be sent must be configured with the following settings:
        - **Request URL** – The endpoint for authentication.
        - **Custom Headers** (optional) – If you want to add and define custom headers to be sent with the request, click **Add Header**. Enter the name and value for the header. Values from the model can be used.
        - **HTTP request type** – Select **POST** or **GET**.
        - **Body Content Type** – The default type is application/json.
        - **Request Body** – Enter the body of the request. Make sure it is in valid format to avoid failed requests.

      Click **Send Request** to send the authentication request.
        - **Authentication Response** – Shows the status of the request, response headers and body.
        - **Authentication Header(s) Passed for Data Sources** (optional) – If you want to pass headers to be sent with Data Source requests, click **Add Header** and specify the following:
            - **Header Text** – Enter the name of the new header for the Data Source requests.
            - **Append Data From** – Define which part of the authentication request response is used as the value. Select **Header**, **Body**, **Cookie**, or **None**.
            - **Input** – Select the correct option from the drop-down menu.

          Click **Add Header** to add another header if needed.
    - If you select **OAuth**, the request to be sent must be configured with the following settings:
        - In the **Parameters for Access Code Request** section, specify the following:
            - **Authorization Endpoint** – The endpoint for authorization.
            - Use **Generated Parameter** fields to define OAuth authorization code flow. Define which parameters are used in the Authorization request based on the target system. Select **Add Parameter** to add more parameters if needed.

      Click **Authorize eShare with OAuth** to authorize eShare with a user account.

- In the **Parameters for Access/Token Refresh Request** section, specify the following:
  - **Token Endpoint** – The endpoint for the request.
  - Use **Generated Parameter** fields to define OAuth authorization code flow. Define which parameters are used in the access token and refresh requests based on the target system. Select **Add Parameter** to add more parameters if needed.

7. Click **Save**.

7.9.11.2. Creating a REST API adapter data source

Add a data source to a REST API adapter to enable the adapter to retrieve specific type of data from the target system. You can define one or more data sources depending on the project needs.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Adapters and Data Sources**. The Adapters and Data Sources configuration view opens.
3. On the Adapters list, click the REST API adapter to which to add the data source.
4. In the Data Sources section of the adapter settings, click **Add data source**.
5. In the Available Data Source Types section, select the data source type to use:
   - **REST API Document Data Source** – to retrieve documents from the external system.
   - **REST API Categorization Data Source** – to retrieve categorization data, for example, to categorize objects according to their construction status (purchased/delivered/installed) or risk level (low/medium/high). The categories have been defined and are maintained in the external system.
   - **REST API Object Attributes Data Source**– To display retrieved data in object attributes in the Model view.
6. Click **Create data source**, and specify the data source-specific settings.
7. Click **Save**.

Related topics

Configuring REST API document data source

Configuring REST API categorization data source

Configuring REST API object attributes data source

7.9.11.3. Configuring REST API document data source

Configure the REST API document data source when you want to retrieve documents and their metadata from the external system. eShare displays the retrieved documents in a document tree in the Documents view, where users can access them. The documents and their metadata are stored and maintained in the target system. If you enable indexing, the system can search for object links in documents, and create links from 3D objects to relevant documents.

When you build the REST API document data source, proceed as follows:

1. Configure the selectors for the root folder or folders.
2. Configure the selectors for the subfolders.
3. Configure the selectors for the documents.

You can define the file types that the adapter supports. The default file types are PDF, DWG, DXF, DGN, and the image types BMP, GIF, JPG, PNG, and TIFF. If the adapter should support other file types, you can specify them according to the project's needs. Note that eShare cannot index these other file types, and the user needs to download the files to be able to view them in an external viewer.

You can work either in basic or advanced mode. In basic mode, eShare helps you build the selectors by providing suitable options in drop-down menus, whereas in the advanced mode, you need to program the selectors in a JavaScript-like but customized selector language. For example, `a.b` means "child b of object a", and `a[3]` means "element 3 of array a".

After you have created a selector, test it by clicking the **Preview** button and analyze the JSON response.

Prerequisites

Create the REST API adapter. Familiarize yourself with the external system and its data structure. You also need to understand the basics of programming.

If you want to enable indexing, ensure that a suitable document handler has been defined. See Adapters and indexing.

Do the following:

1. On the Adapters list, click the REST API adapter to which to add the data source.
2. In the **Data Sources** section of the adapter settings, click **Add data source**.

3. Select **REST API Document Data Source** from the list.
4. Click **Create Data Source**.
5. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
6. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.
    - Remove user groups with the remove button.
    - If the **All Users** group is selected, other group selections will be ignored.
7. The Adapter Configuration section is read-only.
8. In the Data Source Configuration section, specify the following settings:
    - **General**
        - **Mode** – Select the if you want to work in basic or advanced mode.
    - **Indexing and Caching** – Click **Show** to display the configuration.
        - **Indexing** – Select if the document indexing is enabled. If you enable indexing, eShare is able to create links between model objects and documents.

          Note: If you save the data source configuration with indexing, all data sources of the same adapter with indexing enabled, will be indexed again.

          If **Enabled**, specify the following settings:
            - **Indexing Speed**– Define the duration of indexing.

              If you select **Duration**, specify the target duration of each indexing round in **Target Indexing Duration** field (in minutes). The set duration takes effect only after the first full round of indexing has been completed. The default is 120.

              If you select **As Fast As Possible**, the target of each indexing round is to be completed in the shortest time as possible.
            - **Indexing Mode**– Indexing is done every time the data source configuration is saved. Indexing can also be triggered manually from the Project

Administration's General view.

If you select **Once**, indexing is done only after saving.

If you select **Daily**, indexing is done once every day.

If you select **Once per Days of Week**, indexing is done once on the selected days.

If you select **Given Times in Days of Week**, indexing starts on given times of the day on the selected days. The times are given in the server's local time.

If you select **Days of Month**, indexing is done once on the selected days of the month.

If you select **All the Time**, indexing is done continuously. Warning: This can consume a lot of resources if As Fast as Possible is selected as indexing speed.

- **Load Subfolders On Demand** – If enabled, the entire document tree is not loaded at once, but the contents are loaded and visible following the user's examination. The default is **Disabled**.
- **Folder Tree Caching** – Define if the folder tree is cached or retrieved from the server.

  If you select **Disabled**, eShare fetches the folder tree from the server every time the user logs in or refreshes the browser.

  If you select **Enabled**, eShare saves the folder tree in the cache and updates it only periodically. The user sees the cached folder tree. Caching is renewed in intervals defined in **Caching Interval** field. The value is given in minutes.

- **File Types and Hierarchy** – Click **Show** to display the configuration.
  - **Hierarchy Type** – Define how eShare should handle subfolder queries.

    If you select **One-level**, eShare does not query subfolders for further subfolders. Use **One-level** if the target system is simple.

    If you select **Recursive**, eShare will query subfolders for further subfolders. Use **Recursive** if the target system has a complex hierarchy or it is a node-based system.
  - **Ignore File Extensions** – If enabled, filtering documents by file extension is completely disabled. Documents may still be filtered by their file type if their file type is detected in some other manner.

- **File Types** – Select the file types that the document data source adapter should support.
- **Additional File Types to Show**– If the document data source adapter should support other file types, list them here. Separate the file types with a comma.
- **Get Zip file Contents** – Define how eShare should handle ZIP file contents.
  If you select **Disabled**, eShare does not show the contents of the ZIP file in the document tree.
  If you select **Enabled**, eShare shows the contents of the ZIP file to the document tree.
- **File Type Detection** – Define how eShare should detect the file type.
  If you select **Extension only**, eShare determines the document type by its extension, for example, .pdf. If the file does not have an extension, eShare ignores the file. Use this option unless there is a strong reason to select another method.
  If you select **HEAD request**, eShare fetches the document header and attempts to detect the document type from the header. Note that not all systems support HEAD requests.
  If you select **GET request**, eShare downloads the document. GET requests may have an effect on the system's performance, but they are supported by almost all systems. Use GET request only when the other options do not provide the desired result.

- **Custom Request Headers** (optional) – Build custom headers if you need to define custom headers to be sent with the data request; for example, for authorization purposes.
  The header is a name-value pair. The components are target-system-specific, and you can add both static and dynamic elements. Enter the static elements as normal text.

7.9.11.3.1. Configure access to root folders

Root folders are the base folders of the document data source. When you configure the root folders, create first a request that returns information about the root folders. Next, define an array that contains the root folders that you want to use. Preview the response and analyze it. Select the root folder IDs and the root folder names from the JSON response.

Do the following:

1. Click **Show** in the Structure - Root Folders section to display the configuration.
2. Build the URL from which eShare should parse the root folder names and IDs in the **Request URL** field.

   Enter the URL as normal text, but to add the ID, type "ID" and select the ID from the drop-down menu.

   Test the URL by clicking **Test**, and analyze the test response.
3. Create the root folder array selector. In the **Root Folder Array Selector** section, pick the main JSON array where the root folders are listed, and define the array with child elements. The root folder array selector can have one or more child elements. Click **Preview** to verify the result.

   Click **Select a child**, and define the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the root folder array selector by clicking **Preview**. eShare validates the array.
4. Create the root folder ID selector from the array. In the **Root Folder ID Selector 1 From Array** section, select the folder IDs for the root folders. eShare injects the IDs into the request URLs, and links the number of selectors to the number of subfolder ID selectors.

   Click **Select a child**, and define the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the selector by clicking **Preview**.

   To add another ID selector, click **Add ID Selector**.
5. Create the root folder name selector. In the **Root Folder Name Selector From Array** section, select the display names of the root folders. The options depend on the selector.

   Click **Select a child**, and define the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the root folder array selector by clicking **Preview**.
6. To enable getting the documents in the root folder, enter root folder IDs in the **Root Folder IDs** section. Specify root folder ID for each root folder ID selector.

7.9.11.3.2. Configure access to subfolders

eShare fetches the subfolders based on the IDs of the folder on the previous level; either root folders or subfolders. When you configure subfolders, create first a request that returns information about the subfolders. You can use the **Parent Folder ID** as a parameter in the request. Next, define the array that contains the subfolders that you want to use. Preview the response and analyze it. From the JSON response, select the subfolder IDs and the subfolder names.

Do the following:

1. Click **Show** in the Structure - Subfolders section to display the configuration.
2. Build the URL from which eShare should parse the subfolder names and IDs in the **Request URL** field.

   Enter the URL as normal text, but to add the ID, type ID and select the ID from the drop-down menu.

   Test the URL by clicking **Test**, and analyze the test response.
3. Create the subfolder array selector. In the **Subfolder Array Selector** section, pick the main JSON array where the subfolders are listed, and define the array with child elements. The subfolder array selector can have one or more child elements. Click **Preview** to verify the result.

   Click **Select a child**, and define if the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the subfolder array selector by clicking **Preview**. eShare validates the array.
4. Create the subfolder ID selector from the array. In the **Subfolder ID Selector 1 From Array** section, select the folder IDs for the subfolders. The IDs are injected into the request URLs. The number of selectors is linked to the number of subfolder ID selectors.

   Click **Select a child**, and define if the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the selector by clicking **Preview**.

   To add another ID selector, click **Add ID Selector**.
5. Create the subfolder name selector. In the **Subfolder Name Selector From Array** section, select the display names of the subfolders. The options depend on the subfolder array selector.

   Click **Select a child**, and define the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the selector by clicking **Preview**.

7.9.11.3.3. Configure access to documents

eShare recognizes the documents that exist in the subfolder based on the request and array that you create. When you configure the document level, create first a request that returns information about the folder contents. You can use the **Parent Folder ID** as a parameter in the request. Next, select an array that contains the documents in the folder. Preview the response and analyze it. From the JSON response, select the document IDs and document names.

Do the following:

1. Click **Show** in Structure - Documents in Folder section to display the configuration.
2. Build the URL from which eShare should parse the document names and IDs in the **Request URL** field.

   Enter the URL as normal text, but to add the ID, type "ID" and select the ID from the drop-down menu.

   Test the URL by clicking **Test**, and analyze the test response.
3. Create the document array selector. In the **Document Array Selector** section, pick the main JSON array where the documents are listed, and define the array with child elements. The document array selector can have one or more child elements. Click **Preview** to verify the result.

   Click **Select a child**, and define the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the document array selector by clicking **Preview**. eShare validates the array.
4. Create the document ID selector from the array. In the **Document ID Selector 1 From Array** section, select the document IDs. The IDs are injected into the request URLs.

   Click **Select a child**, and define the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the selector by clicking **Preview**.

   To add another ID selector, click **Add ID Selector**.
5. Create the document name selector. In the **Document Name Selector From Array** section, select the display names of the documents. The options depend on the document array selector.

   Click **Select a child**, and define the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the selector by clicking **Preview**.

7.9.11.3.4. Displaying document properties

Define the properties and metadata that eShare needs to display the documents in the Documents view. When you configure the document properties, create first a request that returns information about the document. Build the request using the Document ID parameter. Test the request and analyze the JSON response. Next, select the download ID or link from the JSON response. The response depends on the external service; some services return the full link instead of the ID.

Construct a request that downloads the file using the download ID. Test the request and analyze the JSON response, and select the document name and metadata from the response.

Examples of document metadata that you can display in the Documents view are name, version, author, and ID. You can also create a selector for file types, which will help eShare detect the document type. Name the selector **File Type**, and enter the value in the format "application/pdf".

Do the following:

1. Click **Show** in the Displaying Document Properties section to display the configuration.
2. Select **Make Separate Document Info Request** to be disabled or enabled (default). Disabling the setting enables using Documents in folder request instead of Document info request.
3. Build the URL from which eShare should parse the document names and IDs in the **Request URL** field.

   Enter the URL as normal text, but to add the ID, type ID and select the ID from the drop-down menu. There can be multiple document IDs.

   Test the URL by clicking **Test**, and analyze the test response.
4. Create the document download ID selector. In the **Document Download ID Selector** section, define the array with child elements. The document download ID selector can have one or more child elements.

   Define the child's property name, array index, or a condition that limits the results. Test the document array selector by clicking **Preview**. eShare validates the array. Analyze the information and enter the download URL in the **Download URL** field. This is the URL from where the documents are downloaded.

   To add another ID selector, click **Add ID Selector**.
5. Define the document name selector. This is the property that eShare uses to get the name when downloading the document. In the **Document Name Selector** section, define the array with child elements. The document name selector can have one or more child elements.

   Define the child's property name, array index, or a condition that limits the results. Test the selector by clicking **Preview**.
6. Define the metadata selectors in the **Additional Metadata Selectors** section. eShare uses metadata selectors to provide the user information about the document. If you need to define several similar selectors, you can copy them with the Copy button  and modify them as needed.
   - **Metadata Cache Time** – Configure the cache time for the metadata in minutes.

   To add a metadata selector, click **Add Selector**. Select whether the metadata for the selector comes from **Document Info** or **Folder Info**. Enter the name of the metadata selector. eShare displays the name to the user in the Documents view.

Click **Select a child**, and define the type of the element. Define the child's property name, array index, or a condition that limits the results. Test the document metadata selector by clicking **Preview**.

7. Click **Save**.

7.9.11.4. Configuring REST API categorization data source

Use the REST API categorization data source to categorize objects, for example, based on their status or risk level. For example, in a ship building project, you can categorize the model's objects based on the readiness. The categories have been defined and are maintained in the source system. eShare displays the data in the model.

The REST API categorization data source makes a request to a web server and queries for categorization information. The adapter extracts categorization or model information using JSON queries, and selects items from the response. The adapter maps the extracted data to eShare data using mappings.

Prerequisites

Create the REST API adapter. Familiarize yourself with the target system and its data structure. You also need to understand the basics of programming.

Open the project and the REST API adapter that should have the categorization data source. The REST API adapter is available in the **Project Admin** view, in the **Adapters and Data Sources** section.

Do the following:

1. On the Adapters list, click the REST API adapter to which you want to add the data source.
2. In the **Data Sources** section of the adapter settings, click **Add data source**.
3. Select **REST API Categorization Data Source** from the list.
4. Click **Create Data Source**.
5. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
6. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.

- Remove user groups with the remove button.
- If the **All Users** group is selected, other group selections will be ignored.

7. The Adapter Configuration section is read-only. Continue by configuring the categorization.

7.9.11.4.1. Configuring the categorization

When you configure the categorization, define a query where you specify:

- The main list, or array, of the category/attribute data.
- The data that you need from that list.
- Mappings to define how the data relates to eShare model attributes. Mappings specify how model data is linked to the JSON data provided by the query. Specify one or more correspondences between selectors and the model attributes. The corresponding values of JSON data model object attributes must match, otherwise eShare cannot create the mapping.
- The pieces of data that are relevant for the categorization.

A categorization can have one or more queries.

You can create the query either in basic mode or advanced mode.

- If you use the basic mode, eShare assists you in selecting data.
- In the advanced mode, you need to code the attribute in a custom selector language that is similar to JavaScript. The syntax is as follows:
  - `parent.child`
  - `array[index]`

Test your attribute with the **Preview** button.

If you need create several similar queries or selectors, you can copy them with the Copy button and modify them as needed.

Do the following:

1. **Caching** – This setting defines if data source categorization is always fetched from the data source or if a cached categorization is used.
   - **Disabled** (default) – The data source categorization is always fetched from the data source.
   - **Enabled** – The data source categorization is fetched from the data source when eShare is started or when a data source is configured. After that caching is renewed in intervals defined in **Caching Interval** field. The value is given in minutes.

2. **Visual Style** – When this setting is enabled (default), the user can select the attribute from the visual style drop-down menu, and the 3D view highlights objects with value-specific colors.
    - **Conflict resolution** – This setting specifies which category should override others when multiple categories are defined in this configuration and the database query returns several matches: First Category, Last Category, or the special Multiple Categories category.
3. **Hierarchy** –When this setting is enabled (default), the user can select the attribute from the hierarchy drop-down menu, and the Models tree lists objects in attribute-value-specific nodes.
    - **Conflict resolution** – This setting specifies which category should override others when multiple categories are defined in this configuration and the database query returns several matches: First Category, Last Category, or the special Multiple Categories category.
4. **HTTP request type** — Define if the adapter should be able to read data from or also post data to the service.
    - If you select **POST**, you can specify a body that the system sends with the request. The body is validated while you create it.
    - If you select **GET**, the adapter can only fetch data from the service.
5. **URL** - Build the URL for the external service. The components are target-system-specific, and you can add both static and dynamic elements. Enter static elements of the URL as normal text.
6. Validate the response by clicking **Test**.
7. **Custom headers** (optional) — Build custom headers if you need to define custom headers to be sent with the data request; for example, for authorization purposes.

   The header is a name-value pair. The components are target-system-specific, and you can add both static and dynamic elements.

   Enter the static elements as normal text.

   Test the response by clicking **Preview**. eShare returns the JSON response defined in the query.
8. Continue by creating the query for category data.

7.9.11.4.2. Creating the query

The query consists of the array selector, the data selectors, and the mapping-value selectors.

The array selector selects a JSON array, or list, that provides the categorization data. The data selectors select pieces of data from every element in the array. Mappings connect the data to the actual model data and the eShare categorization system.

When you build the query, proceed as follows:

1. Specify the main array of category (or attribute) data.
2. Specify the data that you need from the array.
3. Specify how the data relates to eShare data by defining the mappings. Also define which pieces of data are relevant for the categorization by using the value selectors.

You can create one or more queries for the REST API categorization data source. If you need to define several similar queries, you can copy them with the Copy button  and modify them as needed.

Do the following:

1. Define the main array of the category data. By default, all data in the defined source system is included in the query, but you can limit it. To limit the data, click **Select a child** and define the list with properties, arrays, or conditions.
2. Create the property selectors. The query can have one or more selectors. This is the actual data that the categorization needs from the main list. Enter the name in the **Name** field, and define the selectors with properties, arrays, and conditions.

   Test the response by clicking **Preview**. eShare returns the JSON response defined in the query. These selectors are available when you create mappings to the model.
3. Add mappings to the model. Click **Add mapping** and create the selector–attribute pairs. Select the selector from the **Selector** drop-down menu and the attribute from the **Attribute** drop-down menu. Attributes have been defined when creating the eShare project.

   Select the value selector from the drop-down menu. The available values depend on the target system. For example, to categorize objects according to their status, select the value that is related to status.
4. Create the categories.
   - **Case sensitivity** – Define if the match should be exact or if text in a different case is accepted. Note that case sensitivity does not apply to range and regular expression values.
   - **Show to user** – If you selected **Listed categories**, the eShare user can see only manually configured categories. If you select **All categories**, eShare creates categories

dynamically for values that are not manually configured, and eShare user can see both dynamically created and manually configured categories.

- **From data** – Creates a table that specifies how the data is categorized. You can sort and refresh the table, and remove unused categories.
    - The **Value** column specifies the value in the target system. eShare categorizes the values that match the given categories; other values are left uncategorized.
    - **Display Value** is the category name that the eShare user can see. If more than one categories have the same display value, eShare merges the categories and they appear as a single category to the user.
    - **Color** – If Visual style is enabled, select the color that you want to use with the color picker. This is the color that eShare uses to represent the elements in that category.
    - **Members** – Displays the number of matching items that, according to the query, exist in the model.
    - **Special categories** – The special categories are **Multiple categories** for objects that meet the criteria of more than one category, and **Uncategorized** for objects that do no meet any criteria.

5. Click **Save**.

7.9.11.5. Configuring REST API object attributes data source

Configure the REST API Object Attributes data source when you want to show data from the source system in the model's object attributes. You can, for example, add information from the project management system to the model. The attributes that you create pick data from the JSON response sent by the external system. You can add object attribute data to the project's model objects or Smart Points.

From the user's perspective, these attributes are just like any other attributes stored in the 3D model, only their values can change dynamically according to data in the external source.

Prerequisites

Create the REST API adapter. Familiarize yourself with the target system and its data structure. You also need to understand the basics of programming.

If you want to add object attribute data for Smart Points, make sure that Smart Point Types have been configured for the model.

Open the project and the REST API adapter that should have the object attributes data source. The REST API adapter is available in the **Project Admin** view, in the **Adapters and Data Sources** section.

Do the following:

1. On the Adapters list, click the REST API adapter to which to add the data source.
2. In the **Data Sources** section of the adapter settings, click **Add data source**.
3. Select **REST API Object Attributes Data Source** from the list.
4. Click **Create Data Source**.
5. In the **Data Source** section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
6. In the **Groups allowed to see the data** section, select user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - Add user groups with **Add**.
    - Remove user groups with the remove button.
    - If the **All Users** group is selected, other group selections will be ignored.
7. The **Adapter Configuration** section is read-only. Continue by configuring the data source.
8. In the **Data Source Configuration** section, specify the following:
    - **Provide data for** – Select whether to retrieve the data to objects or Smart Points.

      If you select **Model Objects**, the retrieved data will be added to objects that have the tag specified in the URL or headers.

      If you select **Smart Points**, the retrieved data will be added to Smart Points of the specified type.
    - **Refresh Automatically**– Define if eShare should refresh the data automatically. If you enable refreshing the data automatically, define the refresh interval in seconds. The default value is 10 seconds.
    - **Allow Search Queries**– Select **Enabled** if the data source can be used in search queries.

      If search queries are enabled, define **Maximum Number of Data Source Queries**, which determines the number of queries the data source can be included in. The field can be left empty for no limit. The default is 250.
9. In **Request Configuration**, specify the following:

- **Request Mode**– Specifies whether incoming data is requested as per object or from a large JSON list.

  If each object or group has its own REST API endpoint, select **Per Object** mode.

  If there is only one endpoint, it is a better option to select **Full List**. If Full List is selected, set the **Keep Cache Time** which specifies how often the list is refreshed. The default is 60 minutes.

  If you want to chain requests, select **Chained Requests**.
- **HTTP request type**– Define if the adapter should be able to read data from or also post data to the service with the request.

  If you select **GET**, the adapter can only fetch data from the service.

  If you select **POST**, you can specify a body that the system sends with the request. The body is validated while you create it.
- **URL**– Build the URL for the target service. The components are target-system-specific, and you can add both static and dynamic elements.

  Enter static elements of the URL as normal text. eShare suggests dynamic elements when you start to enter text; select the element that you want to use.
- **Custom headers** (optional) – Build custom headers if you need to define custom headers to be sent with the data request; for example, for authorization purposes.

  The header is a name-value pair. The components are target-system-specific, and you can add both static and dynamic elements.

  Enter the static elements as normal text. eShare suggests dynamic elements when you start to enter text; select the element that you want to use.
- **Select data for next request**– Visible if selected mode is Chained Requests.
  - Enter **Variable name**.
  - Click **Select a child**.
  - Click **Preview** to verify the result in JSON format.

  To add more data selectors, click **Add data selector**.

  Enter data for second request in the same way.

If you selected **Chained Requests** mode, you can add more requests by selecting **Add request**.

10. In the Testing section, you can test the HTTP request to make sure it provides the correct data. Check that the URL is correct, and click **Test**. eShare provides the JSON response. Analyze the output, and use the results to build the attributes for the data source. The response is target-system-specific.

7.9.11.5.1. Creating the attributes

Create the attributes in the **Response data selectors** section.

The attributes extract data from the JSON response by using selectors. An attribute can map to one or more pieces of data. You can work either in basic mode or advanced mode.

- If you use the basic mode, eShare assists you in selecting data.
- In the advanced mode, you need to code the attribute in a custom selector language that is similar to JavaScript. The syntax is as follows:
  - `parent.child`
  - `array[index]`

If you need to create several similar attributes, you can copy them with the Copy button and modify them as needed. If needed, rearrange the attribute fields with the arrow up and arrow down buttons . You can also build conditions for the attributes.

If you are creating attributes for request per object:

1. Select **Attribute Type**.

   If you select **Text**, do the following:

   - Enter the **Display name**. This is the title of the attribute and displayed in the model object's or Smart Point's properties.
   - Enter the **Display category**. Use the category to group the attributes in the properties pane.

   If you select **Data for Hyperlink**, do the following:

   - Enter the **Display name**. This is the title of the attribute and displayed in the model object's or Smart Point's properties.
2. Build the attribute. Select the data in the **Select** field. In basic mode, the options for selecting data are **Property**, **Array Element**, and **Condition**. The accepted values depend both the selected data type and the external system and are available in the JSON response that eShare provided in the Testing section.
3. To add more variables to the attribute, click **Select a child**.
4. Click **Preview** to verify the result in JSON format.
5. To add more attributes, click **Add attribute**.
6. If you selected Data for Hyperlink as attribute type, in Hyperlinks to show section, specify the following:

- Enter the **Display category**. Use the category to group the attributes in the properties pane.
- Enter the **Link Address**.
- Enter the **Link Display Text**.
- To add another hyperlink, click **Add hyperlink**.

7. To add a document link to the data source, click **Add document link** and specify the following:
   - Select **Document Data Source**.
   - For Document ID Selector, click **Preview**, or, in basic mode, add more attributes by clicking **Select a child**. To add another selector, click **Add ID selector**.
   - For Name Selector, click **Preview**, or in basic mode, add more attributes by clicking **Select a child**.
   - To add another document link, click **Add document link**.
8. Click **Save**.

If you are creating attributes for request for a full list:

Show/hide details

1. In the Array Selector section, the array that contains a list of objects containing attributes is selected. Each object within the array should map to a model object or smart point.
2. Click **Select a child**.
3. Build the attribute. In basic mode, the options for selecting data are **Property**, **Array Element**, and **Condition**. The accepted values depend both the selected data type and the external system and are available in the JSON response that eShare provided in the Testing section.
4. To add more variables to the attribute, click **Select a child**.
5. Click **Preview** to verify the result in JSON format.
6. In the Match Model Attributes/External ID section, select the **Attribute to match**. In basic mode, the options for selecting data are **Property**, **Array Element**, and **Condition**. The accepted values depend both the selected data type and the external system and are available in the JSON response that eShare provided in the Testing section.
7. To add more variables to the attribute, click **Select a child**.
8. Click **Preview** to verify the result in JSON format.
9. To add more attributes, click **Add attribute**.
10. In the Attributes to Show section, enter the **Display name**. This is the title of the attribute and displayed in the model object's or Smart Point's properties.
11. Enter the **Display category**. Use the category to group the attributes in the properties pane.

12. Select **Attribute Type**.

    If you select **Text**, do the following:

    - Enter the **Display name**. This is the title of the attribute and displayed in the model object's or Smart Point's properties.
    - Enter the **Display category**. Use the category to group the attributes in the properties pane.

    If you select **Data for Hyperlink**, do the following:

    - Enter the **Display name**. This is the title of the attribute and displayed in the model object's or Smart Point's properties.
13. To add more variables to the attribute, click **Select a child**.
14. Click **Preview** to verify the result in JSON format.
15. To add more attributes, click **Add attribute**.
16. If you selected Data for Hyperlink as attribute type, in Hyperlinks to show section, specify the following:
    - Enter the **Display category**. Use the category to group the attributes in the properties pane.
    - Enter the **Link Address**.
    - Enter the **Link Display Text**.
    - To add another hyperlink, click **Add hyperlink**.
17. To add a document link to the data source, click **Add document link** and specify the following:
    - Select **Document Data Source**.
    - For Document ID Selector, click **Preview**, or, in basic mode, add more attributes by clicking **Select a child**. To add another selector, click **Add ID selector**.
    - For Name Selector, click **Preview**, or in basic mode, add more attributes by clicking **Select a child**.
    - To add another document link, click **Add document link**.
18. Click **Save**.

7.9.12. SP Foundation adapter

SP Foundation Adapter allows Cadmatic eShare to access Intergraph® SmartPlant® Foundation via a web service, retrieve data, and show the retrieved data as object attributes.

7.9.12.1. Creating an SP Foundation adapter

Create an SP Foundation Adapter to enable Cadmatic eShare to connect to SmartPlant Foundation.

Prerequisites

- You have the host name and address of the SmartPlant Foundation web service.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Open the **Adapters and Data Sources** view.
3. In the Adapters section, click **Add**.
4. In the Available Adapter Types section, select **SP Foundation Adapter**, and then click **Create adapter**.
5. In the Adapter section, enter a name and description (optional) for the adapter, and specify whether the adapter should be enabled.
6. In the Adapter Configuration section, specify the following settings.
    - **SP Foundation Host** – Specify the host name of the SmartPlant Foundation host.
    - **Web Service address** – Specify the address of the SmartPlant Foundation web service.
    - **Username** – Specify the user name for SmartPlant Foundation.
    - **Password** – Specify the password for SmartPlant Foundation.
    - **Project** – Specify the project name, if needed.
    - **Plant** – Specify the plant name.
7. Click **Save**.

Related Actions

- Add a data source to the adapter as described in Creating an SP Foundation adapter data source.

7.9.12.2. Creating an SP Foundation adapter data source

Add a data source to an SP Foundation adapter to enable the adapter to retrieve data from SmartPlant Foundation.

Prerequisites

- The data source can be set up to use either a shared license token for all users or a separate license token for each user. Make sure you are using license tokens according to the End-User License Agreement (EULA) you have made with the provider of the licenses.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Open the **Adapters and Data Sources** view.
3. On the Adapters list, click the SP Foundation adapter to which to add a data source.
4. In the Data Sources section of the adapter settings, click **Add data source**.
5. In the Available Data Source Types section, **SP Foundation Data Source** is already selected – click **Create data source**.
6. In the Data Source section, specify the following settings:
    - **Name** – Enter a name for the data source.
    - **Description** (optional) – Enter a description for the data source.
    - **State** – Specify whether the data source is disabled or enabled (default).
7. In the Allowed for User Groups section, select the **Is Allowed** check box of the user groups that should be allowed to see the data that this data source provides.
    - If no groups are selected, only administrators will see the data.
    - If the All Users group is selected, other group selections will be ignored.
8. The Adapter Configuration section is read-only.
9. In the Data Source Configuration section, specify the following settings.
    - **Use separate token for every user** – If enabled, each user gets a separate token that is valid for 12 hours. If disabled, all users share the same token, which speeds up attribute retrieval.
    - **Comma separated list of supported tags** – Enter the object tags to use in data retrieval as a comma-separated list. For example: sys,pli,nam,vpo,ipo
10. Click **Save**.

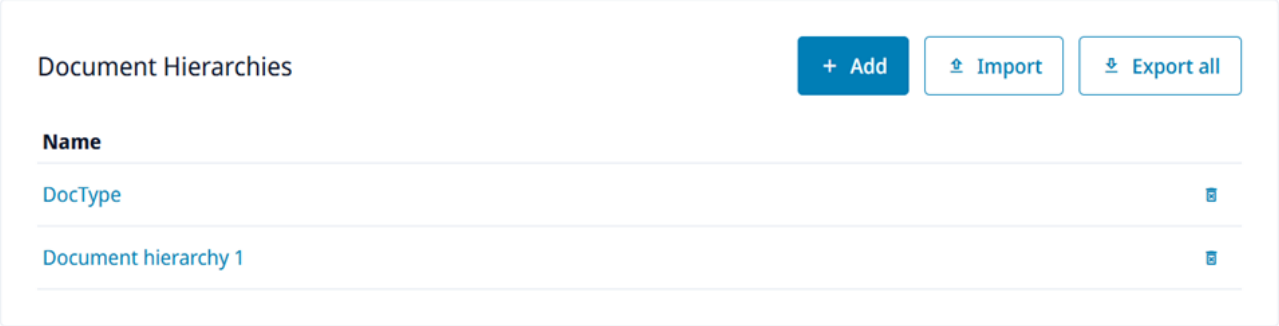
7.10. Managed documents

Cadmatic eShare can be integrated with Cadmatic design applications, which allows project documents such as P&I diagrams to be published from design applications to Cadmatic eShare and then viewed in eShare and eGo. These documents are called "managed documents" and they have a default folder structure that is defined by the application that publishes the document. If a different kind of structure is wanted, project administrator can define a custom document hierarchy. Administrator can define if older revisions of managed documents are saved or if the newer revision always replaces the older revision. Administrator can also choose how links in the documents are highlighted and who can access the documents.

7.10.1. Document hierarchies 217
7.10.2. User groups 219
7.10.3. Link indexing and document revisions 220

7.10.1. Document hierarchies

Project administrator can define custom hierarchies for managed documents published by Cadmatic design applications and export and import document hierarchy configurations.



7.10.1.1. Defining a custom document hierarchy

You can define a custom document hierarchy that determines the kind of folder structure to use for documents published by another application, based on document attributes.

Prerequisites

- A design application has published documents to Cadmatic eShare, and those documents contain attributes.

Cadmatic

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Managed Documents**. The **Managed Documents Configuration** view opens.
3. In the **Document Hierarchies** pane, click **Add**. The **Document Hierarchy Settings** view opens.
4. In the **Name** field, enter a descriptive name for the document hierarchy. This name will be visible to users in the **Documents** view.
5. **Show documents from subfolders** is disabled by default, and only documents stored in the root folder are displayed in the **Documents** view. Enable the setting if you want to display documents also from subfolders.
6. To add a folder level to the document hierarchy, click the **Level 1 Attributes** field and enter the folder attributes as a comma-separated list.

   Each level in the hierarchy definition creates one folder level to the document tree.
   For each attribute that is listed in the first level, each distinct attribute value will create a subfolder.
   For each second-level attribute, each distinct attribute value will create a subfolder in the first-level folders. Example:
   Level 1 Attributes: *A*
   Level 2 Attributes: *B, C*
   If documents have attribute A with values A1, A2, and A3, the first level of the folder structure will have three folders: A1, A2, A3. Then, if the A1 folder contains documents that have attributes B and C with values B1, B2, and C1, folder A1 will have subfolders B1, B2, and C1.
7. To add another level, click **Add New** and specify the required attributes in the same way that you did in the previous step.
8. Click **Save**.

Related Actions

- You can remove a document hierarchy from the **Document Hierarchies** list by clicking button.
- You can reorganize the hierachy levels by dragging them. The levels will be updated automatically. You can delete a hierarchy level by clicking button.
- You can customize an existing document hierarchy by clicking the name of the document hierarchy in **Administration > Managed Documents > Document Hierarchies**.

7.10.1.2. Exporting and importing document hierarchies

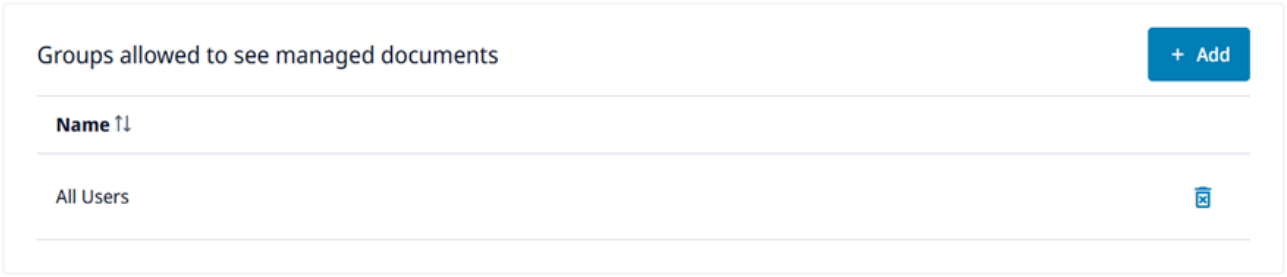
You can export and import document hierarchy settings in text format. You can use this to copy settings from one project to another.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Managed Documents**. The **Managed Documents Configuration** view opens.
3. To export document hierarchy settings, click **Export All**, and then copy the settings from the **Export Document Hierarchies** text box.
4. To import document hierarchy settings, click **Import**, paste the required settings into the **Import Document Hierarchies** text box, and then click **Import**.

7.10.2. User groups

Project administrator can define which user groups are allowed to see managed documents.



Users who belong to a user group that is allowed to see managed documents can do the following:

- See the managed document hierarchy, the documents in the hierarchy, and the metadata of the documents.
- See the links from a 3D object to managed documents when the 3D object is associated with the documents.
- Use the search to find managed documents.

Note: System administrators and project administrators are not affected by these settings—they can always access all managed documents.

7.10.2.1. Defining managed document access rights for user groups

You can specify which user groups are allowed to see managed documents.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Managed Documents**. The **Managed Documents Configuration** view opens.
3. In the **Groups allowed to see the document** section, select **Add**. In the Add user groups dialog, select user groups to add, then select **Add**. To delete a user group, select Delete button for the user group.
4. Click **Save** to save the changes.

Results

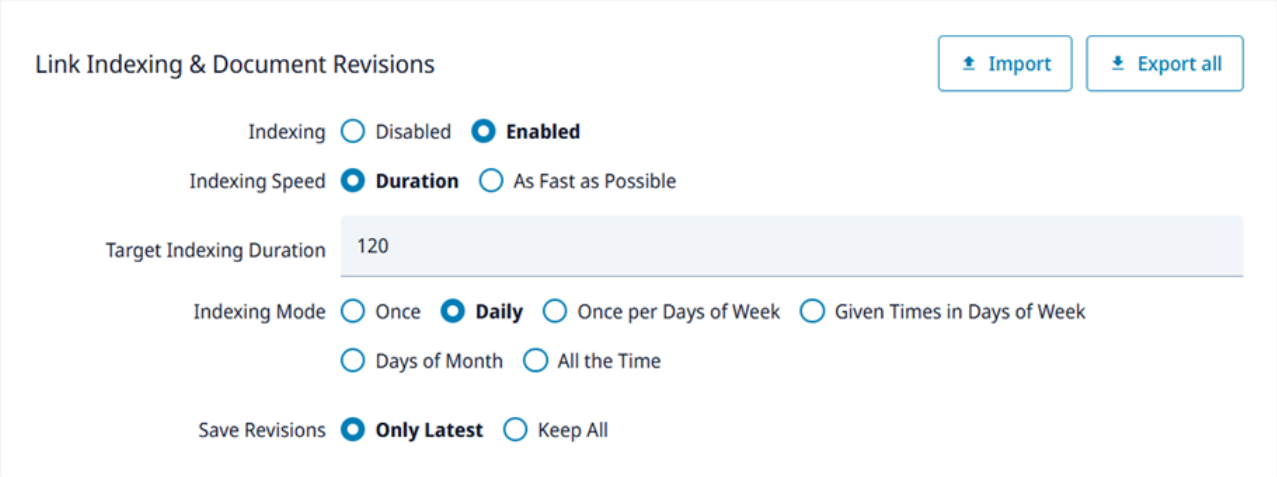
Only the users of the allowed user groups can access the managed documents in eShare and eGo.

Related topics

Project users and groups

7.10.3. Link indexing and document revisions

Project administrator can define if links in managed documents using Document Types are indexed automatically. See Document handlers.



7.10.3.1. Defining link indexing

You can specify whether to enable indexing links in managed documents.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Managed Documents**. The **Managed Documents Configuration** view opens.
3. In the **Link indexing & document revisions** section, select if indexing is enabled for managed documents.

   If **Enabled**, specify the following settings:

   - **Indexing Speed**– Define the duration of indexing.

     If you select **Duration**, specify the target duration of each indexing round in **Target Indexing Duration** field (in minutes). The set duration takes effect only after the first full round of indexing has been completed. The default is 120.

     If you select **As Fast As Possible**, the target of each indexing round is to be completed in the shortest time as possible.
   - **Indexing Mode**– Indexing is done every time the data source configuration is saved. Indexing can also be triggered manually from the Project Administration's General view.

     If you select **Once**, indexing is done only after saving.

     If you select **Daily**, indexing is done once every day.

     If you select **Once per Days of Week**, indexing is done once on the selected days.

     If you select **Given Times in Days of Week**, indexing starts on given times of the day on the selected days. The times are given in the server's local time.

     If you select **Days of Month**, indexing is done once on the selected days of the month.

     If you select **All the Time**, indexing is done continuously. Warning: This can consume a lot of resources if As Fast as Possible is selected as indexing speed.
4. Click **Save** to save the changes.

7.10.3.2. Managing document revisions

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Managed Documents**. The **Managed Documents Configuration** view opens.
3. In **Link indexing & document revisions**, make the following selection for **Save Revisions**:
   - **Only Latest** – The old revision of the existing document is removed from eShare, when a new revision is published.
   - **Keep All** – The old revision of the existing documents is saved in eShare, when a new revision is published.

     The older revisions of the document are accessible in the Document Viewer when the document is open. For details, see Using Documents Viewer. The older revisions are not accessible anywhere else and cannot be found in search.

     Important: If you change the selection from **Keep All** to **Only Latest**, all old revisions of the documents are removed from eShare.
4. Click **Save** to save changes.

7.10.3.3. Exporting and importing link indexing and document revisions settings

You can export and import link indexing and document revisions settings in text format. You can use this to copy settings from one project to another.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Managed Documents**. The **Managed Documents Configuration** view opens.
3. To export document indexing settings, click **Export**, and then copy the settings from the **Export Indexing** text box.
4. To import document indexing settings, click **Import**, paste the required settings into the **Import Indexing** text box, and then click **Import**.

7.11. Document handling

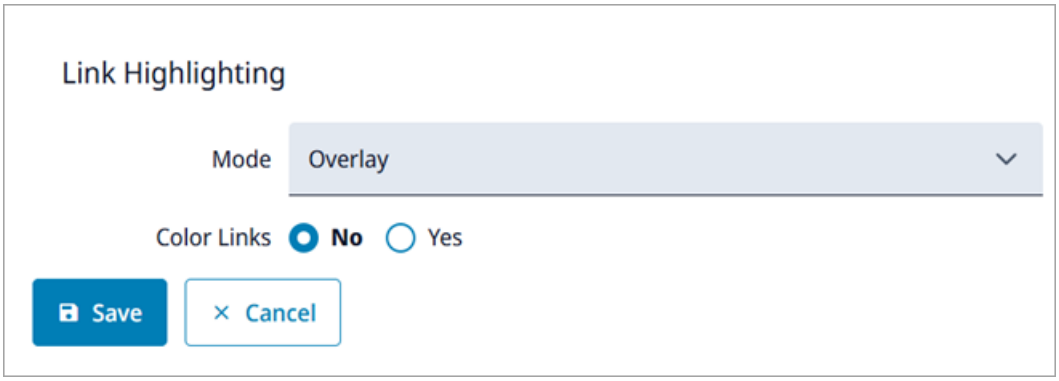
In Cadmatic eShare, users can switch from the 3D view of an object to a PDF document that contains information about that object, or the other way round—from a link in a document to an object in the 3D view.

- Object-to-document links: Cadmatic design tools can publish to eShare a 3D model where the model objects are linked to documents (drawings) published from the same design project. If documents are obtained from an external data source, eShare can create links to model objects if the indexing function is enabled in the data source configuration, as described in Adapters and indexing. Note that any links that eShare adds to Cadmatic documents are not indexed and thus cannot be used for object-to-document linking.
- Document-to-object links: Cadmatic design tools can publish to eShare documents that are linked to the 3D model published from the same design project. The document viewer of eShare displays these links, and project administrator can customize link highlighting, as described in Link highlighting. If, however, the Cadmatic documents do not contain such links, or documents are obtained from an external data source, in the Document Handling view a project administrator can define rules and patterns that enable eShare to detect text strings and convert them into hyperlinks. When a user opens a document and document handling settings for that document type exist, eShare automatically creates links that the user can click to open the relevant 3D view.
- OCR configuration : Use Optical Character Recognition to extract data from documents in eShare, as described in OCR configuration (beta version).

7.11.1. Link highlighting 223
7.11.2. Document handlers 227
7.11.3. Document conversion formats 237
7.11.4. Linking multiline text strings 238
7.11.5. OCR configuration (beta version) 239
7.11.6. Exporting and importing document handling settings 241

7.11.1. Link highlighting

Project administrator can define if how links are highlighted in managed documents.



7.11.1.1. Defining link highlighting

You can specify whether to highlight the links to model objects in managed documents. If the links are highlighted, a specific coloring method can be selected for indicating something about the target of the link; for example, the link color can indicate that the target object is not found in the 3D model, or that the object has a specific Status Tracking value.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Open the **Document Handling** view.
3. In the **Default Links for Managed Documents** section, select **Modify**.
4. In the **Link Highlighting** section, specify these settings:
    - **Mode** – Select the mode to use. If the project contains documents that have a large number of links, some highlighting modes can slow down the speed of browsing the documents, and you should choose the mode based on the expected performance.

Mode	Description	Expected Performance
Overlay	Links have a transparent overlay.	Slowest
Underline	Links are underlined.	Faster
Borderline	Links have a border around them.	Faster
Mixed	Highlighting varies, depending on what link type has been assigned to each link in the	Normal

Mode	Description	Expected Performance
	application that publishes the document: ◦ Symbol links have overlay. ◦ Label Text links are underlined. ◦ Default type links are underlined if they fit to a single line, otherwise they have overlay.	
None	Links have no special indication, but they are still active.	Fastest

- Color Links – If set to Yes, in the Link Coloring Configurations you can select links to be colored based on a specific property of the target objects.

  If set to No, the highlight color is always blue.
- In Link Coloring Configurations, specify the following settings:

  Name – Give the coloring configuration a descriptive name. This name is shown also in Document Viewer.

  ◦ Color According to

  If set to Model , select a color from the color picker for:

  ■ Object Found
  ■ Object not Found
  ■ Attribute not Found
  ■ Show Legend – Set to Yes if you want a status color legend to be displayed in the documents.

  For information on configuring model attributes, see Attribute settings.

  If set to Status Tracking, specify these settings:

  ■ Used Status Trackers – Select a status tracker from the list.
  ■ No Category – Select a color from the color picker.
  ■ Object not Found – Select a color from the color picker.
  ■ Show Legend – Set to Yes if you want a status color legend to be displayed in the documents.

For more information on status tracking, see Status tracking and object grouping.

If set to **Model Attribute Categorization**, specify these settings:

- **Used categorization** – Select a categorization from the list. You can add multiple categorizations.
- **No Category** – Select a color from the color picker.
- **Object not Found** – Select a color from the color picker.
- **Show Legend** – Set to **Yes** if you want a status color legend to be displayed in the documents.

For more information on model attribute categorizations, see Attribute categorization.

If set to **Data Source Categorization**, specify these settings:

- **Used categorization** – Select a categorization from the list. You can add multiple categorizations.
- **No Category** – Select a color from the color picker.
- **Object not Found** – Select a color from the color picker.
- **Show Legend** – Set to **Yes** if you want a status color legend to be displayed in the documents.

For more information on data source categorizations, see the documentation of the relevant adapter in Adapters and data sources.

If set to **Smart Point**, specify these settings:

- **Used Smart Point Types** – Select a smart point type from the list. You can add multiple types.
- **No Category** – Select a color from the color picker.
- **Object not Found** – Select a color from the color picker.
- **Show Legend** – Set to **Yes** if you want a status color legend to be displayed in the documents.

For more information on Smart Point types, see Smart Point types.

To add a new coloring configuration, click **Add**.

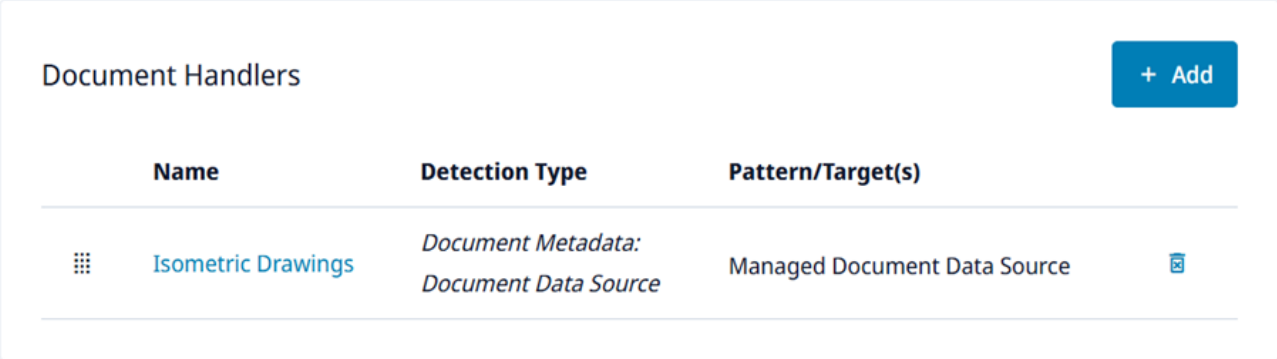
5. Click **Save** to save the changes.

7.11.2. Document handlers

Document handler defines how Cadmatic eShare can turn text labels in a document into links that the user can click to open the referenced entities in the 3D viewer.

If all documents refer to model objects in a consistent manner, defining a single document handler might be enough. Multiple definitions are needed for example when two document handlers contain text labels that have identical formatting but different purpose (label "1234" refers to System in one document type but to Sub-block in another), or when different document handlers require the target ID of links to be queried from a different attribute.

If the Document Handlers list contains more than one document handler, you can rearrange the document handlers by dragging.



7.11.2.1. About document metadata

Before defining a document handler configuration for documents obtained from an external system, project administrator must define the document metadata that will be used to add links to the documents. This is done in the document data source settings of the adapter that Cadmatic eShare uses to connect to the external system.

If using a database document data source, the first selected value in the Document File Query is always used as the name of the document. This, and all other values in the query, are provided to the document parser as metadata fields.

In general, metadata fields are key–value pairs, where the key is a column name or alias from the Document File Query, and the value is the value of that column or alias. When entering the name of such metadata field into a document handler configuration, make sure you type it correctly.

Cadmatic

Related topics

Adapters and data sources

7.11.2.2. About document type detection

To enable Cadmatic eShare to add links to different document types, project administrator must define how document type can be detected from document metadata. This metadata might be stored in various locations: in a document attribute, in systematically assigned drawing numbers, or in a keyword in the document title. When the location is known, project administrator can configure the document adapter to store the document type information in a metadata field. This metadata field can then be used in the document handler configuration, as described in Document handlers.

When a user selects to open a document, eShare applies the document type definitions from top to bottom, looking for a match. Accordingly, the topmost definitions should match specific document types, and the last definition should match any remaining document type, so that all documents are mapped to a document type.

It is also possible to create a single document handler definition that matches all documents:

- If all documents are of the same type, you can use a metadata field that has a fixed value.
- If there are several document yypes and type is stored in a metadata field, in the detection rule you can use a regular expression that matches all possible values of that metadata field.

7.11.2.3. About document processing rules

Document processing rules define which parts of a text string are converted into links, and how the IDs of the target items are formed. The simplest way to do this is to use the automatic detection rules. This compares every text element in the documents with known object and group IDs, and when it finds a match it links the text to the entity in the 3D model.

Automatic detection cannot be used in the following situations:

- If IDs in document labels and in the 3D model are not identically formed. When this is the case, identifier detection rules can be used to detect the different spellings and convert them to match the IDs in the 3D model.
- If IDs in document labels are not complete. For example, a part of the ID is omitted from the document because it is the same for all items in the document. When this is the case, identifier detection rules can be used to extract the missing part from a metadata field and add it to all IDs, so that they match the IDs in the 3D model.

- If false matches occur because document contains text that looks like an ID but is not, and you know that this kind of ID should not be present in this document. When this is the case, you can create rules that only match valid IDs, and false matches are ignored. Note that it might not be possible to eliminate all false matches; for example, if IDs consist of digits only, any numbers in dimension labels can be detected as IDs.
- If IDs in document labels do not exist in the model, but can be found from external data sources. For example, a P&ID might contain IDs of equipment and pipes that have not been modeled yet, but which already exist in the P&ID database and in other systems. The document handler rules do not check whether the target ID exists in the model; an ID in the document only needs to match the formatting defined in the rule for a link to be created, and if there is an external data source to provide for it, the user can access it from the link.
- If the text in the document labels is split to two or more rows (more than one character per row).

When automatic detection cannot be used, project administrator must specify how the system can detect the required text strings by configuring manual rules for detection.

You can mix document handlers that use automatic detection with those that use specific rules.

Here is an example of how document processing rules can be used:

- In the document, find a text string that starts with the character "V" followed by three digits.
- Combine the text string with the value of the "System" document attribute.
- Find the combined value from the "Valve Position ID" attribute of model objects.

Using this rule set, if document metadata contains the attribute System=1301 and the document contains a text label with the string "V402", the text label is linked to valve whose position ID in the 3D model is "1301V402".

7.11.2.4. Defining a document handler

In the Document Handling view, the Document Handler settings allow project administrator to define how Cadmatic eShare should identify and process documents obtained from Cadmatic design applications or from an external data source. Document handler specific document processing rules enable eShare to convert text labels into links, so that users can click a label to jump from a drawing to the related entity in the 3D view.

Prerequisites

- A document adapter defines a document data source that contains documents. File System Document Adapter is the simplest method—it can access documents stored in the local file system or a mapped network drive. See Adapters and data sources.
- To define Metadata Processing Rules, the metadata field has been defined in the document data source settings of the adapter.
- To highlight links using colors that represent part's status, the required status tracking settings have been defined. See Status tracking and object grouping.
- Document conversion format settings have been defined, if needed. See Document conversion formats.
- You know how to use regular expressions. See About .NET regular expressions.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Open the **Document Handling** view.
3. In the Document Handlers pane, click **Add**. The Document Handler Configuration view opens for defining the following settings.
4. **Settings**
    - **Name** – Enter a descriptive name for the document handler. Example: "Isometric Drawings".
    - **Target Document**
        - **External** – Documents are obtained from some other system or application. If selected, specify the following settings:
            - **Conversion format** – **Default Format** keeps the format of the original document. To use a custom format defined in document conversion

settings, select it from the list.

- **Force paper size in conversion** – Select **No** to keep the page size of the original document. Select **Yes** to convert documents to the page size defined in the selected document conversion format.
- **Use OCR** – To use OCR with managed documents, select **Yes** and select the OCR configuration from the list.

- **Managed** – Documents are published from Cadmatic design applications. If selected, specify the following settings:
  - **Show Managed Links** – Set to **Yes**, if you want managed links to be visible. Default is **Yes**.
  - **Use OCR** – If OCR has been enabled and configured, to use OCR with managed documents, select **Yes** and select the OCR configuration from the list. See OCR configuration (beta version).

- **Show Managed Links** – Set to **Yes**, if you want managed links to be visible. Default is **Yes**. Selection is visible, if **Target Document** is set as **Managed**.

5. **Document Type Identification**

   If selected Target Document is **External**, select the detection method:

   If both detection methods are enabled, both methods will be used when determining the document handler. The opened document only uses this configuration, if it is from the listed data source, and it contains the specified metadata key with a matching pattern.

   - **With Metadata** – If enabled, specify the following:
     - **Identifier Name** – Enter the name of the metadata field to use for detecting document type. Enter "Path" to detect it from the file path.
       If the configuration targets managed documents, a drop-down menu of suggested identifier names is shown.
     - **Pattern to Match** – Enter a regular expression that Identifier Name must match to designate document type. If the value is a fixed string, just enter the string. Or, define a more complex expression, for example if only a part of the metadata field's contents can be used for detecting document type.
   - **With Data Source(s)** – If enabled, specify the following setting:
     - **Target Document Data Sources** – Add a document data source. There can be multiple data sources.

> **Note:** All the documents under the listed data sources will use the configuration. If the configuration targets managed documents and detection method is set to **Data Source**, the configuration will target all the managed documents in the project.

If selected Target Document is **Managed**, specify the following:

- **With Metadata** – If enabled, specify the following:
  - **Identifier Name** – Enter the name of the metadata field to use for detecting document type. Enter "Path" to detect it from the file path.
    If the configuration targets managed documents, a drop-down menu of suggested identifier names is shown.
  - **Pattern to Match** – Enter a regular expression that Identifier Name must match to designate document type. If the value is a fixed string, just enter the string. Or, define a more complex expression, for example if only a part of the metadata field's contents can be used for detecting document handler.

6. **Metadata** – Specify how to link text labels in a document to objects in the 3D model.
   - **Processing of Existing Metadata** define which document metadata fields are to be used for constructing links. To add a field, click **Add new** and specify the following settings:
     - **Source Field** – Enter the name of the metadata field to use. Make sure you type the name correctly.
     - **Processing** – Select whether to use the metadata field's value as-is, or to extract only a part of the value. If you choose the latter option, an additional field is displayed; in this field, enter a .NET regular expression that extracts the required value and contains named capture groups, and then use those groups in the Target ID Format field of the Link Creation Rules section.
   - **Extract Metadata from Documents**– If set to **Yes**, a preview window with a document affected by the document handler configuration appears. To change the document used in the preview, modify the configuration in Document Type Identification section.
   - **Metadata Extraction Areas** – Shown when metadata extraction enabled.
     To add metadata extraction areas to the document, select **Add new**. A new rectangular area with two crosses in opposite corners appears in the preview window. The area can be moved by dragging the crosses. The new rule is also added below the preview window.

- **Pattern** specifies which parts of the text within the metadata extraction area should be extracted, and what they should be called. The pattern should be a regular expression with at least one named capture group, marked with <> notation. All of the named capture groups can be used in Identifier Detection Rules to generate links. By default, a rule is assigned a pattern that extracts all text in the area within a single named capture group.
- **Origin Corner** defines which corner the metadata area anchors itself to. This means that the area will always stay at the same distance from the corner, regardless of document size. By default, the origin corner is set as Bottom Left.
- **Page** defines which page of the opened document the metadata area will be used on. When a new rule is created, its page is automatically assigned to be the page number currently open in the preview document. The document preview only shows metadata areas assigned to that page. The value can be a negative number. In this case, the page of the metadata area will be counted from the end of the document. For example, page value set as -1 refers to the last page of the document, -2 to the second to last, and so on.

To test the metadata extraction rules on the preview document, select **Test Extraction Rules**. The test outputs a table with the metadata keys on the left (capture group names), and the extracted results on the right.

7. **Document Link Coloring**
   - **Color Links** – If set to **Yes**, in the Link Coloring Configurations you can select links to be colored based on a specific property of the target objects.
   - In **Link Coloring Configurations**, specify the following settings:
     - **Name** – Give the coloring configuration a descriptive name. This name is shown also in Document Viewer.
     - **Color According to**

       If set to **Model**, select a color from the color picker for:
       - **Object Found**
       - **Object not Found**
       - **Attribute not Found**
       - **Show Legend** – Set to **Yes** if you want a status color legend to be displayed in the documents.

       For information on configuring model attributes, see Attribute settings.

If set to **Status Tracking**, specify these settings:

- **Used Status Trackers** – Select a status tracker from the list.
- **No Category** – Select a color from the color picker.
- **Object not Found** – Select a color from the color picker.
- **Show Legend** – Set to **Yes** if you want a status color legend to be displayed in the documents.

For more information on status tracking, see Status tracking and object grouping.

If set to **Model Attribute Categorization**, specify these settings:

- **Used categorization** – Select a categorization from the list. You can add multiple categorizations.
- **No Category** – Select a color from the color picker.
- **Object not Found** – Select a color from the color picker.
- **Show Legend** – Set to **Yes** if you want a status color legend to be displayed in the documents.

For more information on model attribute categorizations, see Attribute categorization.

If set to **Data Source Categorization**, specify these settings:

- **Used categorization** – Select a categorization from the list. You can add multiple categorizations.
- **No Category** – Select a color from the color picker.
- **Object not Found** – Select a color from the color picker.
- **Show Legend** – Set to **Yes** if you want a status color legend to be displayed in the documents.

For more information on data source categorizations, see the documentation of the relevant adapter in Adapters and data sources.

If set to **Smart Point**, specify these settings:

- **Used Smart Point Types** – Select a smart point type from the list. You can add multiple types.
- **No Category** – Select a color from the color picker.
- **Object not Found** – Select a color from the color picker.
- **Show Legend** – Set to **Yes** if you want a status color legend to be displayed in the documents.

For more information on Smart Point types, see Smart Point types.

To add a new coloring configuration, click **Add new**.

8. **Link Creation Rules** enable the system to detect text that matches a given pattern and link the text to the related entity in the 3D model. When the user opens a document that contains text strings, the first identifier detection rule that matches the text is used. Accordingly, the topmost rules should identify some special cases, and these would be followed by more general rules, so that all required texts can be converted into links.

   If text labels consist of multiple lines, see Linking multiline text strings.

   - **Automatic detection of model objects** and **Automatic detection of smart points**.

     The automatic detection of smart points compares the document to the external IDs of smart points.

     Select the check box for the rule(s) to use. The configuration is shown when the rule is enabled (check box selected). Click **Show Options** for each rule to specify the following settings:

     - **Maximum Combined Line Length** – Sets the maximum length of components that are used to define a label. The default value is 3. Increasing this can help find additional labels, but it may affect performance. When the field is left empty, the default value is used.
     - **Line Item Search Range** – Sets the relative size of the area used to search for components of a label. The default value is 2. Increasing this can help find additional labels, but it can affect performance. When the field is left empty, the default value is used.
     - **Maximum Line Number** – Defines the number of lines in a document that can be combined to a single link. The default is 2. When the field is left empty, the default value is used.

   - **Pattern rules** – Configuration is displayed when the rule is enabled (check box selected).

     Note: Cadmatic AI can be used to assist in creating a configuration. Select **Cadmatic AI** in the menu to open chat. See Cadmatic AI.

     To create a new rule, click **Add new** and specify the following settings:

     - **Pattern rule** – Enter a name for the pattern rule or use the default name. The field cannot be empty.

- **Pattern to Match** – Enter a .NET regular expression that can capture the required text strings. By using named capture groups in the pattern, you can use the whole captured text or just parts of it, and then use the groups in the Target ID Format field.
- **Target ID Format** – Specify how the ID of the target object or group in the 3D model is formed. You can use the following elements:
  - Metadata fields or parts of them, as defined in the Metadata Processing Rules section.
  - Named capture groups from the Pattern to Match field.
  - Text strings.
- **Target Type** – Select whether the target in the 3D model is:
  - **Any model item** – The target can be any type of item. The attribute of the item must be either its key attribute or a group defining attribute.
  - **Model items identified with an attribute** – The target can be only items whose identifier is a specific attribute. If this is selected, an additional **Attribute** field is displayed, and you can select the attribute or its *abbreviation* from the drop-down menu. The attribute of the item can be either a key attribute, a group defining attribute, or a normal attribute.
  - **Smart Points** – The target is Smart Points. If this is selected, an additional **Smart Point Type** field is displayed, and the used Smart Point Type must be selected. The target type must have an External ID attribute in use.
- Click **Show Options**, if you want to specify the following settings for the pattern rule:
  - **Maximum Combined Line Length** – Sets the maximum length of components that are used to define a label. The default value is 3. Increasing this can help find additional labels, but it may affect performance. When the field is left empty, the default value is used.
  - **Line Item Search Range** – Sets the relative size of the area used to search for components of a label. The default value is 2. Increasing this can help find additional labels, but it can affect performance. When the field is left empty, the default value is used.

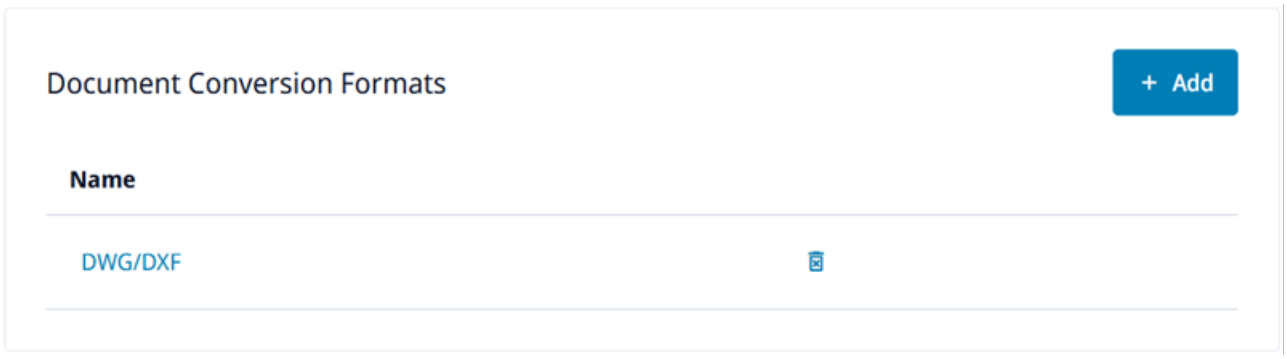
Click **Show Options**, if you want to specify the following settings for the document processing rules:

- **Skip Pages From Start** – If you set a positive value to the field, the link injection will skip the set number of pages from the start, such as title page or revision information. The default is 0, which means no pages are skipped. When the field is left empty, the default value is used.
- **Skip Pages From End** – If you set a positive value to the field, the link injection will skip the set number of pages from the end. The default is 0, which means no pages are skipped. When the field is left empty, the default value is used.

9. Click **Save**.
10. To apply the settings, publish the 3D model.

7.11.3. Document conversion formats

Document conversion format settings enable project administrator to customize how Cadmatic eShare converts DGN, DWG, and DXF files into the PDF format, for example in regard to paper size and orientation.



7.11.3.1. Defining a document conversion format

Perform the following to define new document conversion format settings.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Open the **Document Handling** view.
3. In the **Document Conversion Formats** pane, click **Add**. The Document Conversion Format Settings view opens.
4. In the **Document Conversion Settings** pane, define these settings:

- **Name** – Enter a name for the settings. Example: "DWG/DXF".
- **Paper Size** – Select paper size. The smallest size is A4, and the largest is 8A0 (eight times A0).

  Note: Document handler settings define whether to use original page size or this value.

- **Orientation** – Select **Landscape** or **Portrait**.
- **Color Scheme** – Select **Black & White**, **Grayscale**, or **Full Color**.
- **Convert Pages** – Select whether to convert only **Active** pages or **All** pages.

5. Click **Save**.

   You can now select this conversion format in the Conversion format settings of a document handler.

Related topics

Document handlers

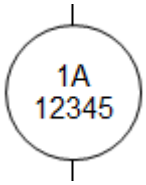
7.11.4. Linking multiline text strings

Normally, Cadmatic eShare expects a text string that should be turned into a link to be a single line of text, because in PDF documents each line of text is handled as a separate string.

There are two exceptions to this:

- When a text label has multiple lines but just one character per line, as in when label text is in Japanese, it can be handled as a single string.
- When a text label consists of two lines that are close enough to each other and horizontally aligned to left, middle, or right. This must be set up in the document type definition by describing the format of the first line and the second line separately, and using *\n* to represent the line break in the middle.

For example, a diagram drawing might have labels in the following format, so that the first line always consists of a number followed by a letter, but depending on object type the second line might consist of numbers or letters:



To detect such labels, the Identifier Detection Rule should look for the pattern *<number><letter><line break><five numbers or letters>*, and the rule could be constructed as follows:

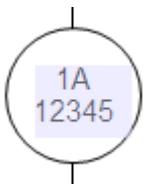


This example pattern creates two named capture groups, "LineOne" and "LineTwo", and uses two character classes: the decimal digit *\d* character class that only matches numbers, and the word character *\w* character class that matches both numbers and letters.

- *(?<LineOne>\d\w)* creates the named capture group LineOne that represents any string that consists of one decimal digit and one word character. If the first line had one number and two letters, you could use *\d\w\w*.
- *\n* stands for the line break.
- *(?<LineTwo>\w{5})* creates the named capture group LineTwo that represents any string that consists of five consecutive word characters.

The **Target ID Format** field specifies that the first part of the string must match the pattern that LineOne represents, and the second part must match the pattern that LineTwo represents.

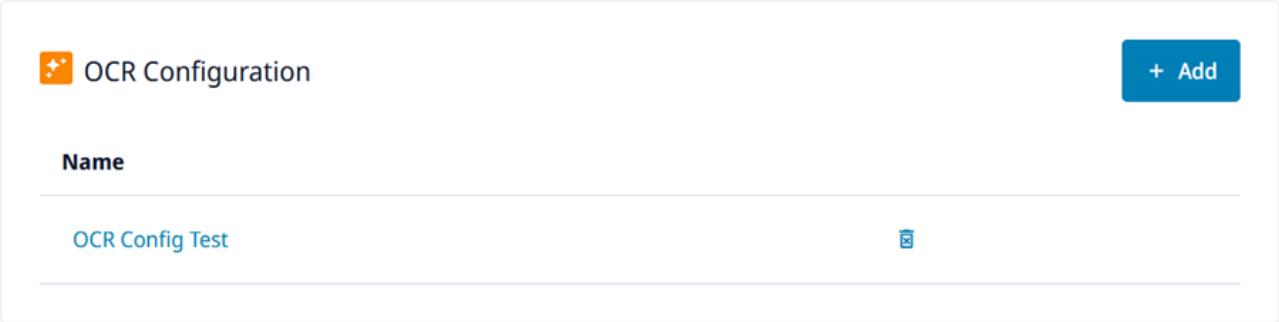
When the user opens a document from Cadmatic eShare and a multiline label text matches a pattern, the system turns the text into a link that the user can click to open the related object in the 3D view.



7.11.5. OCR configuration (beta version)

Optical Character Recognition (OCR) is a technology which converts text in images or scanned documents into machine-readable and searchable text. OCR enables systems to recognize letters,

numbers, and symbols within visual content, making it possible to search, copy, edit, and index thr extracted text for further use. OCR can now be used in eShare to extract data from documents.



7.11.5.1. Configuring OCR

Prerequisites

- OCR is enabled in eShare.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Open the **Document Handling** view.
3. In the **OCR Configuration** pane, click **Add**. The OCR Configuration Settings view opens.
4. In the configuration settings, specify the following:
    - **Name** – Enter a descriptive name for the configuration.
    - **Execution** – Specifies when OCR is run for documents. To run OCR when document is being opened, select **On document open**, or to run OCR when indexing is done, select **Only on index**.
    - **On collisions prioritize** – Specifies which extraction should be prioritized in scenarios where there are detections from OCR and also in embedded text in the PDF. In this case the user has initially loaded the document without OCR and data on it already exists. If the OCR is ran after this, there could be detections where current data already exists. Select either **OCR** or **Embedded text** to set which is prioritized.
    - **Prioritize on extract** – Specifies which should be prioritized, if the document contains text strings in any format. If you know that the contents contain correct embedded data, select **Embedded text**, but if the embedded data is incorrect, select OCR.
5. Click **Save**.

7.11.5.2. Attaching OCR to document handlers

After OCR has been configured, you can select to use OCR when you create a new document handler or edit an existing handler. The option **Use OCR** is available in the configuration settings. Set the option to **Yes** and select the OCR configuration on the list. See Defining a document handler.

7.11.5.3. Using OCR with documents

When OCR has been configured and enabled for a document handler, it will be active the next time you open or index a document based on the configuration. When OCR process is run, a text and a spinner with the Cadmatic AI logo are shown. If the OCR process gets stuck or you want to abort it, click the **Cancel** button on the document load page. The data from OCR is cached to the document database. This means the OCR process has to be run only once; subsequent requests load the data directly from the database, significantly improving loading performance. OCR processing can be time-consuming, especially for larger documents. It is recommended to run it during indexing.

If text extraction fails and you want to try again, you can clear the cached document data in **Project Admin > General** for the document data source. See Viewing the indexing status .

7.11.6. Exporting and importing document handling settings

You can export and import document handling settings in text format. You can use this to copy settings from one project to another.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Document Handling**. The **Document Handling** view opens.
3. To export settings, click **Export All**, and then copy the settings from the **Export Document Types** text box.
4. To import settings, click **Import**, paste the required settings into the **Import Document Types** text box, and then click **Import**.

7.12. Project users and groups

System administrators are automatically also project administrators—they can access and administrate all projects. They can add and remove project users, and set project administrator permissions to project users.

Project administrators can add and remove project users, but they cannot set project administrator permissions.

Project users can be individual eShare users or eShare user groups, and project administrator permissions can be set to both types. That is, an eShare user can be a project user or project administrator by being designated as one, or by being a member of a group designated as one.

User lists can be transferred from one project to another via export/import.

Note: You can also use a command-line tool to add users to a project. See Command-line interface for administrative tasks for details.

7.12.1. Adding and removing project users and groups 242
7.12.2. Exporting all project users and groups 245
7.12.3. Importing project users and groups 245

7.12.1. Adding and removing project users and groups

In the Project Users and Groups view, users who have project administrator permissions can add and remove project users and project user groups, and system administrators can define whether a specific user or user group has project administrator permissions.

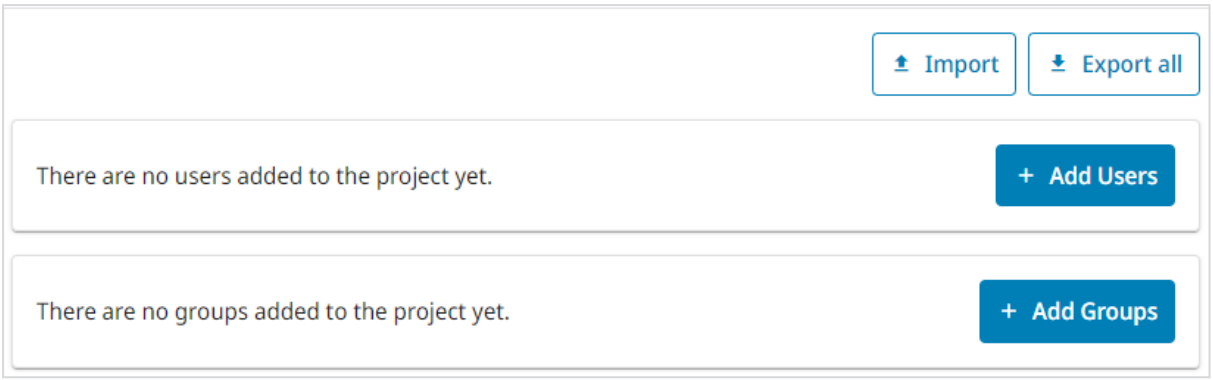
Adding a user group allows all the eShare users in that group to access the project. Setting a user group as project administrator gives project administrator permissions to every user in that group.

Prerequisites

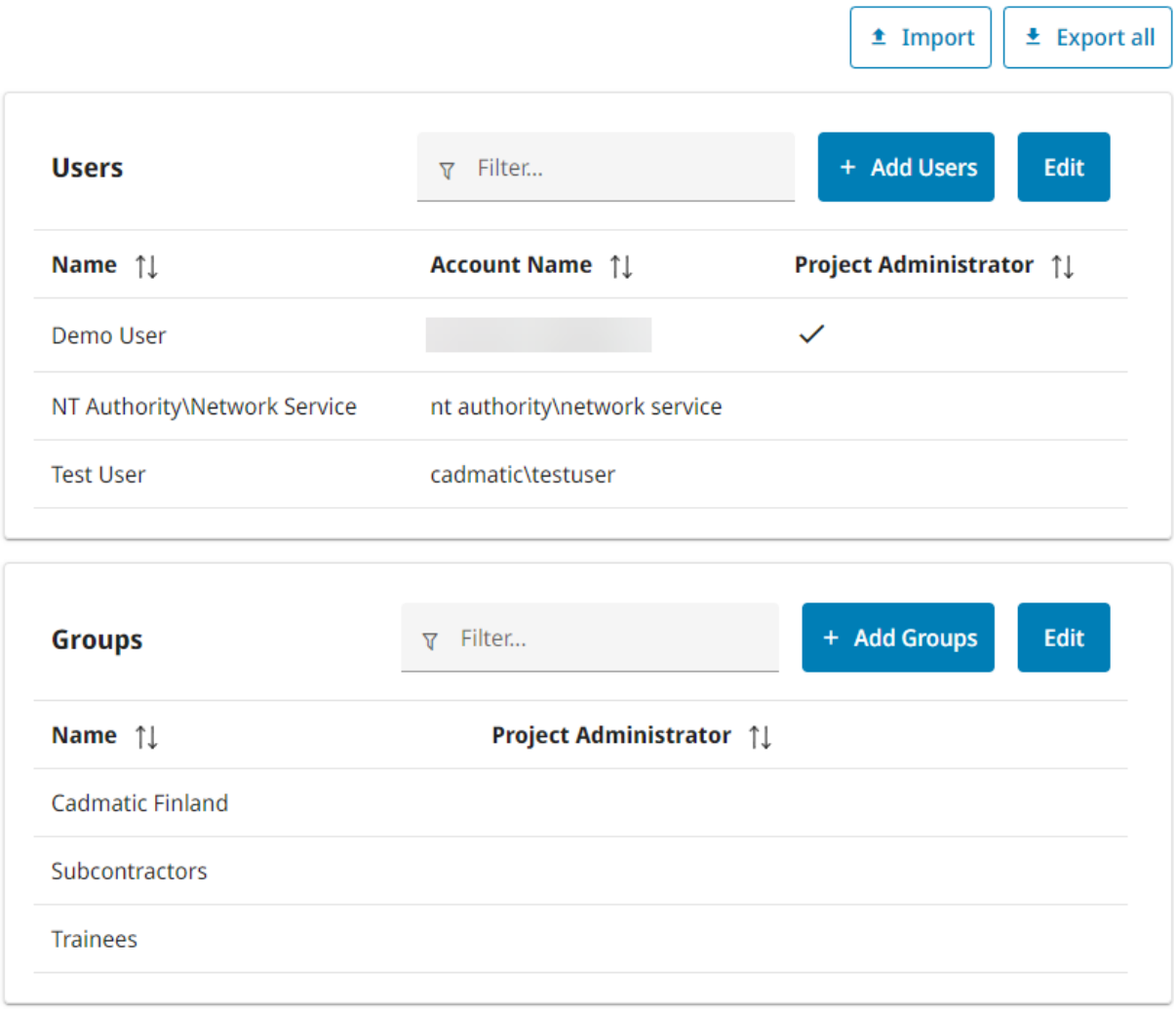
- The user or user group to add to the project exists in Cadmatic eShare. See Users and Groups.
- To set project administrator permissions, you must be a system administrator.

Do the following:

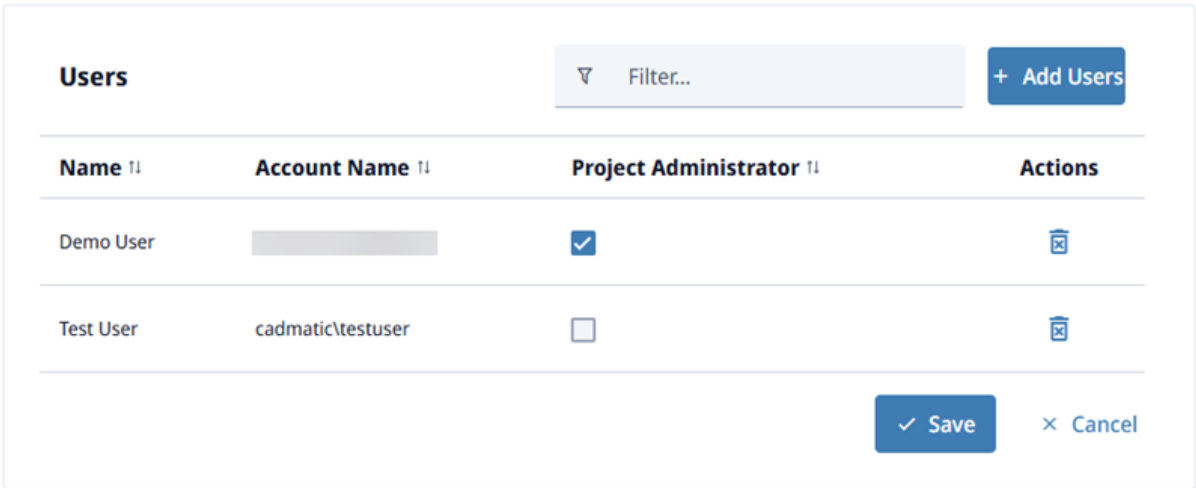
1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Users and Groups**. The Project Users and Groups view opens.



3. To add a user to the project, select **Add Users**. **Add Users** dialog opens. Select user(s) to add, and then select **Add**. (**Add Users** is hidden if all users have already been added to this project.)
4. To add a user group to the project, select **Add Groups**. Add Groups dialog opens. Select group (s) to add, and then select **Add**. (**Add Groups** is hidden if all groups have already been added to this project.)



5. To remove a user or user group from the project, select **Edit** for users or groups, and then Delete button for the user or group.



6. (System Administrator) To set the project administrator permissions of a user or user group, select **Edit** and then select the **Project Administrator** check box of that user.

Note: Typically, you might not want to give project administrator permissions to a group, but only to individual users who need these permissions. You do not need to give project administrator permissions to users who are system administrators, because they are permitted to administrate all projects.

Results

The designated users and user groups are permitted to access the project, either as project administrators or normal users.

7.12.2. Exporting all project users and groups

You can export all project users and groups in text format. You can use this to copy users from one project to another, or just to back up the user list.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Users and Groups**. The **Project Users and Groups** view opens.
3. In the **Project Users and Groups** pane, click **Export All**. The **Export Project Users and Groups** text box displays the settings.
4. Select all settings and copy them to the clipboard. You can save the settings in a text file if needed.

7.12.3. Importing project users and groups

You can import a previously exported user configuration to a project.

Prerequisites

- You have a user configuration export on the clipboard. See Exporting All Project Users and Groups.
- The users to import are already Cadmatic eShare users. See Users.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Users and Groups**. The Project Users and Groups view opens.
3. In the Project Users and Groups pane, click **Import**. The Import Project Users and Groups text box is displayed.
4. Paste the configuration to import into the text box.
5. Select whether to import project administrator permissions.
    - **Yes** – Users that have project administrator status in the imported configuration will also have project administrator status in the target project.
    - **No** – All users are imported as normal project users.
6. Click **Import**.

7.13. Project colors

In the Project Colors view, the project administrator can define a color palette for the project. The colors set in Project Colors view are used in all the color pickers for the project in eShare.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Colors**. The Project Colors view opens, listing the currently defined colors.
3. Click **Edit**. The fields of existing project colors become editable.
4. To add a new color to the project colors, click **Add Color**, and then enter the following information.
    - **Name** – Enter a name for the color. Spaces and special characters are allowed in the name.
    - **Value** – Set a value for the color or use the color picker.
5. To edit an existing project color, edit its name or value as appropriate.
6. To change the order of the colors, drag the colors to the desired order.
7. To delete a project color, click the delete button of that color.
8. Click **Save**.

7.14. Project variables

In the Project Variables view, project administrator can define variables that store a text string such as file path or web address. These variables can then be used in adapter and data source configurations. The view also has a section for setting the AI context for the project and the configuration of AI data sharing.

7.14.1. Setting project variables

Each project has two built-in variables: {projectName} and {projectDbName}.

- {projectName} – Contains the name of the project.
- {projectDbName} – Contains the name of the project database.

7.14.1.1. Adding, editing, and deleting project variables

When an adapter or data source configuration refers to a project variable, any changes made to the variable will not be automatically applied to those configurations. If you rename a variable, you must manually correct its name in the configurations that refer to it. If you edit the value of a variable, you must open and save each configuration that refers to the variable, to apply the change. If you delete a variable, you must manually remove it from configurations that refer to it.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Variables**. The Project Variables view opens, listing the currently defined variables.
3. Click **Edit**. The fields of existing project variables become editable.
4. To add a new project variable, click **Add Variable**, and then enter the following information.
    - **Name** – Enter a name for the variable. Spaces and special characters are allowed in the name.
    - **Value** – Enter the value of the variable.
5. To edit an existing project variable, edit its name or value as appropriate.
6. To delete a project variable, click the delete button of that variable.
7. Click **Save**.

Results

You can use the currently defined project variables in adapter and data source configurations by entering the variable name in curly brackets.

For example, if the variable is called "DWG", you can refer to it with *{DWG}*.

7.14.1.2. Exporting project variables

You can export all project variables in text format. You can use this to copy variables from one project to another, or just to back up the variable list.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Variables**. The Project Variables view opens.
3. In the Project Variables pane, click **Export All**. The Export Project Variables text box displays the settings.
4. Select all settings and copy them to the clipboard. You can save the settings in a text file if needed.
5. Click **Close**.

7.14.1.3. Importing project variables

You can import previously exported project variables to a project.

Import removes all existing project variables. When an adapter or data source configuration refers to a project variable, any changes made to the variable will not be automatically applied to those configurations. If import changes the value of a previously known variable, you must open and save each configuration that refers to the variable, to apply the change. If import permanently deletes a variable, you must manually remove it from configurations that refer to it.

Prerequisites

- You have a project variable export on the clipboard. See Exporting project variables.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Variables**. The Project Variables view opens.

3. In the Project Variables pane, click **Import**. The Import Project Variables text box is displayed.
4. Paste the configuration to import into the text box.
5. Click **Import**.

Results

Existing project variables are removed, and the imported project variables are displayed on the list.

7.14.2. Setting AI context

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Variables**. The Project Variables view opens.
3. In the AI context pane, click **Edit**. The context field opens for editing and you can enter the project's AI context.
4. Click **Save**.

7.14.3. Setting permissions for AI data sharing

When AI data sharing is enabled, Cadmatic may review the AI assistant conversations in the project, including the project data the assistant looked up, to improve the assistant. By default, AI data sharing is disabled, and the conversations are stored only to run the chat and are never reviewed for this purpose. When a user deletes a chat, it is removed from the review data as well.

Do the following:

1. Navigate to the project to edit, and then click **Project Admin** in the main menu. The project administration view opens.
2. Click **Project Variables**. The Project Variables view opens.
3. In the AI data sharing pane, enable or disable **Share this project's AI chats with Cadmatic** option.

   By default the option is disabled.

8. System administration

In Cadmatic eShare a system administrator can manage projects, users, and user groups.

- To open system administration, select **System Admin** in the menu.



8.1. Dashboard 250
8.2. Projects 254
8.3. Users 259
8.4. Groups 266
8.5. Devices 269
8.6. Licenses 272
8.7. JSON Web Tokens 274
8.8. External services 274

8.1. Dashboard

In **System Administration > Dashboard** a system administrator can view details on the application pool, see background tasks in more detail and view log messages.

- To open the Dashboard view, select **Dashboard** in the menu.



In **Application Pool** pane the administrator can see the user identity of the application pool and the running time since last startup.

Background Tasks pane shows the number of projects currently publishing, status of point cloud synchronization and the indexing status of document data sources and managed documents. Click **See details** to view the information on each project in the system in more detail.

Latest Server Log Messages pane shows most recent error messages and information on parameter configurations. Click **Logging settings** to configure logging targets. Click **Refresh** to update the log. **See details** to view more detailed information and make changes to log settings.

8.1.1. Viewing the indexing status of document data sources and managed documents

Do the following:

1. Click **System Admin** in the main menu. The Dashboard opens.
2. In the **Background Tasks** pane, click **See details**.

3. See **Document indexing** section for the indexing status of document data sources and managed documents.
4. You can make the following selections:
    - Click the name of the data source to see its data source configuration. **InternalDocuments** are managed documents.

      Note: If you save the data source configuration, all data sources of the same adapter with indexing enabled, will be indexed again.

    - To clear cached document data, click . This will delete the cached data and the text extraction will be ran again. This can be useful in cases where the text extraction has failed and you want to try again. The text extraction should run every time the document is updated, but this can also be forced by clicking the button.
    - To clear the indexing results, click . This will remove all the indexing results from the database, nullifying all indexing of the data source.
    - To restart the indexing, click . The indexing will be restarted and completed using the indexing speed set in the data source configuration.
    - To restart the indexing and complete it as fast as possible, click . The indexing will be restarted and completed as fast as possible by forcing the data source to use As Fast as Possible as the indexing speed.

8.1.2. Configuring logging settings

Do the following:

1. Click **System Admin** in the main menu. The Dashboard opens.
2. In **Latest server log messages** pane, click **Logging settings**.
   In Logging settings view all the logging targets are listed. The default logging target is eShare server log which cannot be deleted and its type cannot be changed from *Log file*.

8.1.2.1. Adding a logging target

Do the following:

1. Click **System Admin** in the main menu. The Dashboard opens.
2. In **Latest server log messages** pane, click **Logging settings**.
3. In **Other logging targets** pane, select **Add logging target**.

4. Specify the following:
    - **Type** – Select the type of the logging target. The options are **Elasticsearch**, **JSON file**, or **Log file**.
    - **Filters** – Control which log events are captured.
        - **Message logging**– Set the log level of the log messages. The options are **Trace**, **Debug**, **Information**, **Warning**, **Error**, **Critical**, and **None**.
        - **Performance logging**– Specify to include the performance of certain features in the log. The options are:
            - **Publishing** – Logs publish steps and how long each step took, including submodel generation.
            - **Indexing** – Logs each step of indexing in the background: who initiated indexing and why, and how long indexing a document took.
            - **Opening documents** – Logs how long it took for eShare to open a document.
            - **Opening model** – Logs how long it took for eShare to open the model.
        - **User activity logging**– Specify which user activities are included in the log. The options are:
            - **Session start & end** – Logs when users logged in and out, including their IP address and if login failed.
            - **License reservation & release** – Logs information about license failures, and how many licenses were acquired.
    - If you selected **Elasticsearch**, specify the following:
        - **URI** – The endpoint of the Elasticsearch cluster.
        - **ILM policy** – The name of the ILM policy to use.
        - **Base64 API key** – Base64-encoded API key for auithenticating with Elasticsearch.

          Note: Using Elasticsearch requires authenticating using either Base64 API key or user name and password.

        - **User name** – User name for authenticating with Elasticsearch.
        - **Password** – Password for authenticating with Elasticsearch.
    - If you selected **JSON file** or **Log file**, specify the following:
        - **Folder name** – The directory where the log file is stored. The folder path is relative to the log folder defined in Application settings for Internet Information

Services (IIS) Manager. See Configuring application settings.

- **File name** – The name of the log file. The file will be created inside the folder specified in **Folder name**.
- **Time** – Specify the time used for the timestamps in the log entries. The options are **Local time** or **UTC**.

5. Select **Save**.

8.1.2.2. Editing a logging target

Do the following:

1. Click **System Admin** in the main menu. The Dashboard opens.
2. In **Latest server log messages** pane, click **Logging settings**.
3. Select **Edit** for the logging target either in **eShare server log** or **Other logging targets** pane. The logging target settings open.
4. Make needed modifications.

   Note: The default logging target is eShare server log and its type cannot be changed.

5. Select **Save**.

8.1.2.3. Deleting a logging target

Note: The default logging target is eShare server log which cannot be deleted.

Do the following:

1. Click **System Admin** in the main menu. The Dashboard opens.
2. In **Latest server log messages** pane, click **Logging settings**.
3. Select **Delete** for the logging target in **Other logging targets** pane.
4. Select **Delete** in the confirmation dialog.

8.1.3. Viewing and downloading the log

Do the following:

1. Click **System Admin** in the main menu. The Dashboard opens.
2. In the **Latest Server Log Messages** pane, click **See details**.

3. To view a log from an earlier date, click the arrow by the date and select the date from the drop-down menu.
4. The log view can be filtered using the columns **Timestamp**, **Type**, **Message**, **Source**, or **Thread ID**.
5. To download the log, click **Download**.

8.1.4. Changing log settings

Do the following:

1. Click **System Admin** in the main menu. The Dashboard opens.
2. In the **Latest Server Log Messages** pane, click **See details**.
3. In **eShare Log Feed** view, click **Filters** to define included columns in the drop-down menu. To reset the columns and filters, click **Reset columns and filters**.

8.2. Projects

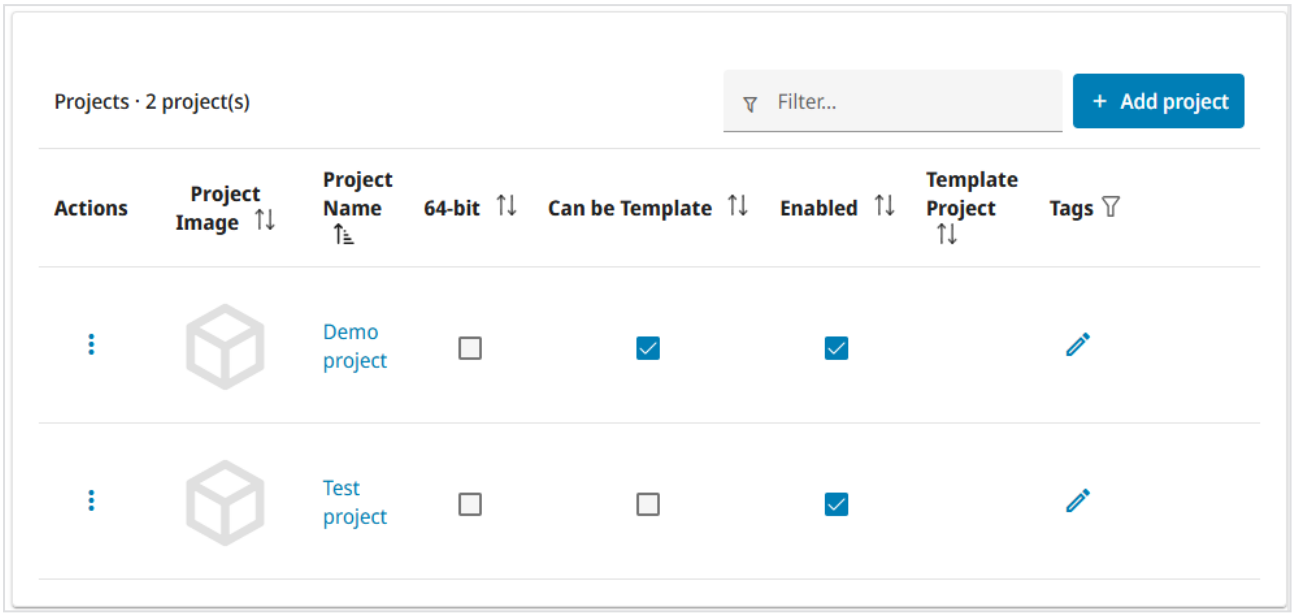
In **System Admin > Projects** a system administrator can create and delete projects.

8.2.1. Project management view

In the Project Management view of Cadmatic eShare a system administrator can add and delete projects.

- To open the Project Management view, select **Projects** in the menu.





The title row of the Projects list indicates how many projects there are in total. Click **Add project** to add a new project, as described Adding a new project.

In project-specific rows of the project list you can do the following:

- Click ⋮ in Actions column and select:
  - **Edit** to edit project details. See Editing project details.
  - **Delete** to delete the project. See Deleting a project.
- **Project Name** – Click the project name to open the project administration view. See Project administration.
- **64-bit** – Select to specify that the project is so large that it must be opened in eShare App instead of a web browser.
- **Can be Template** – Select to specify that the project can be used as a template for new projects.
- **Enabled** – When a project is enabled, licensed users and applications are able to access the project and exchange data with it. For example:
  - Authenticated eShare project administrators and users can see and access the project in eShare oreShare App.
  - Cadmatic Plant Modeller can publish 3D models and documents to the project.
  - Cadmatic P&ID can retrieve object information from the project.
  - Cadmatic eGo can download the project and synchronize project data with eShare.

Clearing the **Enabled** check box disables the project immediately. You can disable projects that you do not want any user or application to be able to access.

You might also want to disable old, non-active projects, because this can improve the start-up time of the server. When a project is disabled, the server does not load the related adapters and data sources, index the documents, or synchronize the point cloud files.

Note: Cadmatic eGo users who downloaded the project before it was disabled can continue using their offline copy, but they will not be able to synchronize the data with eShare.

- **Template Project** – Shows the name of the template project if the project was created from a template.
- **Tags** – Click to select an existing project tag or add a new project tag. Click to to confirm changes.

  You can also filter the project list based on tags. There are two selections in the filtering menu:

  - **Includes** – Selection for tags, which the projects must have. It will include all the projects that have all of the selected tags. For example, selecting tags *one* and *two* means that projects which have both of these tags will be shown, so for example *one, two* or *one, two, three* will be visible, but *one, three* will not.
  - **Excludes** – Selection for tags, which the project cannot have. It will exclude all the projects that have any of the selected tags. For example, selecting tags *one* and *two* means that it will not show projects that have either of these tags. So for example projects with tags *one, three* or *two*, or *one, two*, will not be visible.

  The filters can then be either cleared or applied from the footer of the filter menu. These filters are saved locally.

8.2.2. Adding a new project

Cadmatic Plant Modeller can be configured to create projects in Cadmatic eShare, but if your site does not work this way, a system administrator must manually create the required design projects in Cadmatic eShare as described below. Each project is stored in a separate database.

Prerequisites

- You are a system administrator.

- The database account that Cadmatic eShare uses is allowed to create databases. If this is not the case, a database administrator must create the project database as described in *Cadmatic eShare Database Administration Guide* before you can start creating the project.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Projects**. The project management view opens.
3. Click **Add project**. The Project configuration view opens.
4. Enter the following information.
    - **Project Name** – Enter a name for the project. This is the name that users will see.
    - **Database Name** – Enter a unique name for the project database, using only alphanumeric characters and underscores.

      If a database administrator has already created the database, enter the name provided by the administrator.

      If you leave the field blank, the system will generate a name that begins with *ESharedProject_*.
    - **Template Project** – You can select to use an existing project as a template. If you select a template project, the new project will get complete configuration details from the template project, but this does not copy the 3D model or any documents from the template project. This is the same as using the project configuration import described in Importing project configuration, except that when using the Template Project option the properties of the default Markup Type are always copied to the new project whereas in project configuration import it is optional.

      Note: Any changes made to the template project after this, including deleting the template project, will not affect the new project you are creating.

    - **64-bit (Allow eShare App Only)** – If set to **Yes**, the project can only be opened in eShare App which allows running Cadmatic eShare in a 64-bit environment. eShare App should be used for very large projects, but it also provides improved visualization in smaller projects.
    - **Can be Template** – If set to **Yes**, the project can be selected as Template Project for other projects. If set to **No**, it is still possible to export the project configuration for import to another project.
    - **Image** – You can add an optional image for the project.

- **Description** – You can add an optional description for the project.
- **Tags**– Select from existing project tags in the drop-down list or create a new one. A project can have multiple tags.
- **Coordinates** – You add the coordinates for latitude and longitude for the project location. If map support is enabled, you can use **Select from map** to open project map. To enable map support, see Map support.

  The first time the map is opened, it prompts for location permission. If enabled, it will initialize the location at your current location, or if the project has a location already that will be used instead. Otherwise a predetermined location is used. You can select a location from the map by clicking, which updates the coordinates and is visualized with a marker on the map. To close the map, select **Close map**.

5. Click **Save**. The new project is added to the Projects list. If you chose to use a template project, its name is displayed in gray.

Related topics

General

Project users and groups

8.2.3. Deleting a project

You can delete a project that is no longer needed in Cadmatic eShare. This deletes the database of that project and the project configuration. The operation cannot be undone.

Prerequisites

- You are a system administrator.
- Database administrator has made a backup of the project database if needed.
- The database account that Cadmatic eShare uses is allowed to delete databases. If this is not the case, make a note of the name of the database, and have database administrator delete the project database after you have deleted the project, as described in *Cadmatic eShare Database Administration Guide*.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Projects**. The project management view opens.
3. On the list of projects, locate the one to delete, and then click the delete button of that row. You are prompted to confirm the action.

4. If Cadmatic eShare could not delete the project database, ask your database administrator to do it.

8.3. Users

In **System Admin > Users** a system administrator can add and remove users, and edit user details.

Basic user authentication is based on Microsoft Windows user accounts, and the users can be defined in a domain or workgroup. But before a Windows user cansign in to eShare, an eShare system administrator must add the Windows user account to eShare and define the user as active. When adding a user to eShare, the user type is defined as system administrator or normal user:

- **System administrators** can access all views, all projects, and all data. They can perform system-level configuration actions such as add and activate other users, define user groups, and create and delete projects.
- **Normal users** can only access data that relates to projects to which they have been added. When adding a user to a project, the user role is defined as project administrator or user:
    - **Project administrators** can add and remove other project users, edit project properties, upload and publish 3D models, and configure adapters that enable the project to exchange data with other systems.
    - **Project users** can access and annotate the 3D model and view project documents.

To further limit users' access to specific data, system administrators can create user groups and assign users to these groups. When a project uses data adapters to exchange data with other systems, a project administrator can specify which user groups are allowed to see project data obtained from a specific data source.

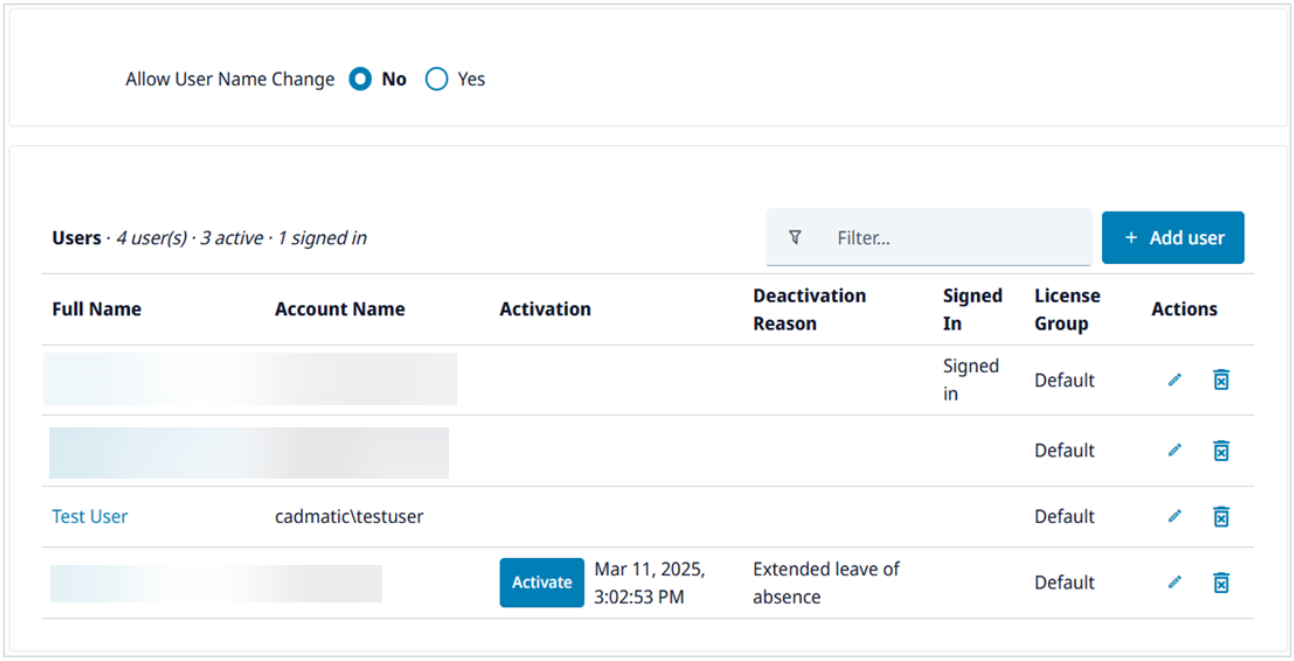
To summarize, a Cadmatic eShare user can be a system administrator with complete administrative rights to everything or a normal user who has either administrative or user rights to specific projects, and user permissions can have additional limitations through use of user groups. For information on managing eShare users, see the following topics.

8.3.1. User management view

In the User Management view of Cadmatic eShare a system administrator can manage users' ability to access the system.

- To open the User Management view, select **Users** in the menu.





Allow User Name Change – If set to **Yes**, project users can change their visible user name in their user profile view. By default the setting is disabled. See My Account.

The title row of the Users list indicates how many users there are in total and how many of those are active. (Here "active" refers to *the ability to sign in* to the system—it does not necessarily mean that the users are actually signed in and using the system.)

You can click **Add user** to add a new user to the system; see Adding a new user for details.

You can use the filter field to show fewer users on the list.

In user-specific rows of the user list you can do the following:

- Click the edit button to view or edit user details. See Editing user details.
- Click **Activate** to activate a deactivated user account. See User activation and deactivation.
- Click the delete button to delete the user account. See Deleting a user.

8.3.2. Adding a new user

A system administrator can add new users to Cadmatic eShare to allow them to sign in and use the system.

- To allow a Microsoft Windows user to sign in to Cadmatic eShare, add the user account of that user.
- To allow a Cadmatic design application to publish content to Cadmatic eShare, add the user account that the publishing service uses. For more information on publishing, see Integration with design applications.

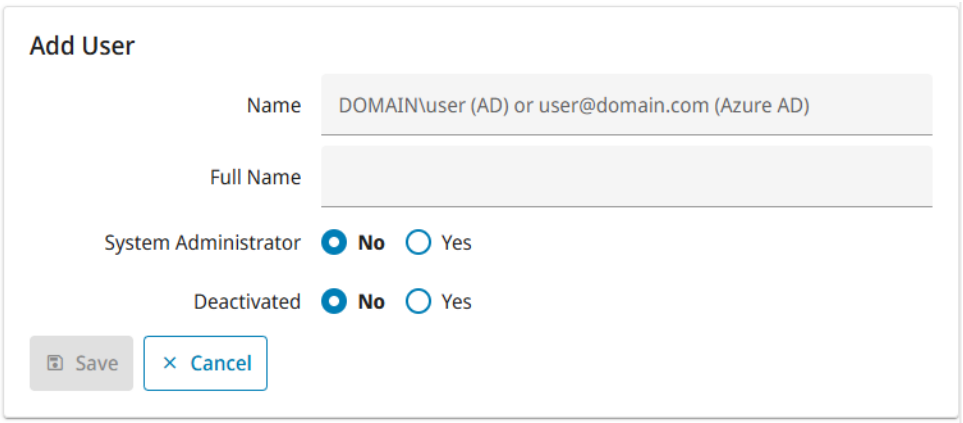
Note: Instead of defining users manually as described below, you can also use command-line tools to import users from a domain user group. See Command-line interface for administrative tasks for details.

Prerequisites

- You are a system administrator.
- The user to add is defined in Active Directory or in a Windows Workgroup.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Users**. The User management view opens, listing the users currently defined in the system.
3. Click **Add user**. The User Details view opens.



4. In the **Name** field, enter the username.
   If it is an Active Directory domain user, enter it as *DOMAIN\user*.

If it is an Azure Active Directory domain user, enter it as *user@domain.com*.

5. In the **Full Name** field, enter the username to be displayed in Cadmatic eShare.

Note: The system does not enforce Full Name to be unique, but if several users have the same name it can be difficult in some views to select the appropriate user. You can also edit the name after creating the user.

---

6. **System Administrator** is set to **No** by default, to add the user as a normal user. If you set it to **Yes**, the user will have system administrator permissions (which includes project administrator rights in all projects).
7. **Deactivated** is set to **No** by default, to allow the user to sign in and use the system. If you set it to **Yes**, the user will not be able to sign in, and the **Deactivation Reason** field is displayed for entering an optional comment about why the user is deactivated.
8. Click **Save**.

Results

The user account is added to Cadmatic eShare.

If the user account was activated and there is an available license, the user can open a web browser and sign in to the system.

Project administrators can assign the user to projects, as described in Adding and removing project users and groups.

8.3.3. Editing user details

A system administrator can edit user details such as full name and system administrator status. You cannot remove system administrator status from yourself or deactivate yourself.

Note: If you are about to edit user details because you want to deactivate a user, see Deactivating a user instead.

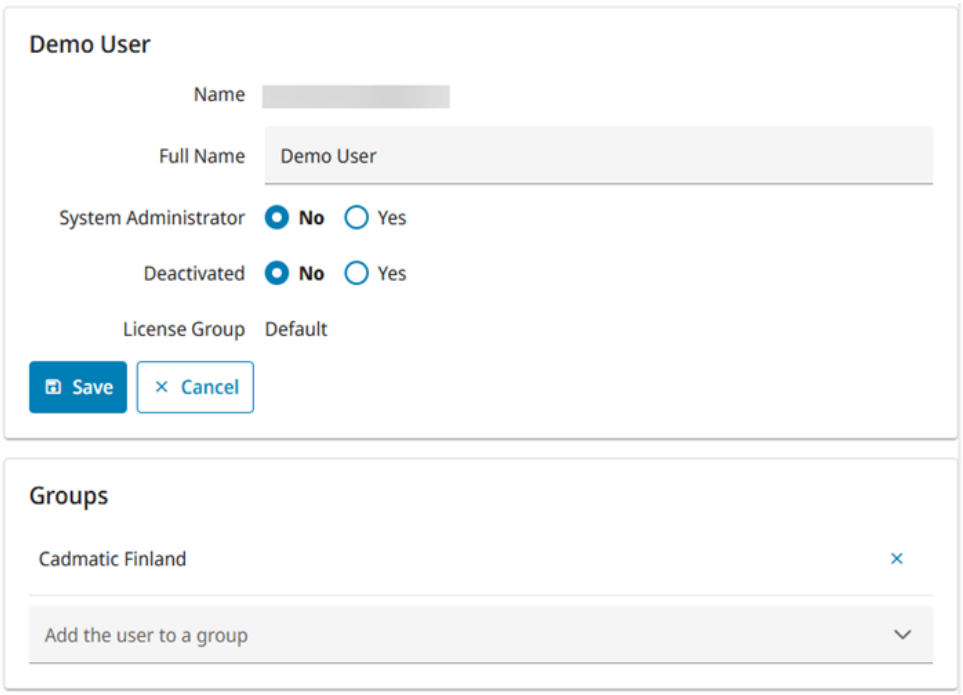
Prerequisites

- You are a system administrator.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Users**. The User management view opens.
3. On the list of users:

- Click the Full name or the edit button . A view with user details opens.
- Click **Edit** to edit the details.



4. Change the details as required.
5. Click **Save**.
6. In the **Groups** section, you can add the user to a groups by selecting the group from the list, or remove from a group with the remove button .

Results

The changes are applied to the user.

8.3.4. User activation and deactivation

System administrator can activate and deactivate users to specify whether they are allowed to access Cadmatic eShare.

8.3.4.1. Activating a user

System administrator can activate a user whose account either the system or a system administrator has deactivated. An activated user consumes a user license, either immediately or at the latest when the user signs in to the system.

Prerequisites

- You are a system administrator.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Users**. The User management view opens.
3. On the list of users, you can click **Activate** to immediately activate the user account.

   But, if you want to review or edit user settings before the activation, perform the remaining steps.
4. Click the name of the user to edit. The User Details view opens.
5. Click **Edit**.
6. Set **Deactivated** to **No**, and change any other settings if appropriate.
7. Click **Save**.

Results

The user account is activated, and the user can sign in to the system.

8.3.4.2. Deactivating a user

User deactivation prevents the user from signing in to Cadmatic eShare. It does not delete the user account or remove the user from the user list of any project that the user has been assigned to, and deactivated users can still be added to projects.

Deactivation can occur automatically if the system detects that there are not enough licenses for all users. System administrators can deactivate users manually if there is a need to release licenses or prevent access to the system. You cannot deactivate your own user account.

Deactivation is immediate. Users who are signed in when they become deactivated are informed that due to deactivation they can no longer use eShare. Database transactions that were initiated before the deactivation occurred are completed, but any unsaved data that the user might have entered in the user interface is lost.

Prerequisites

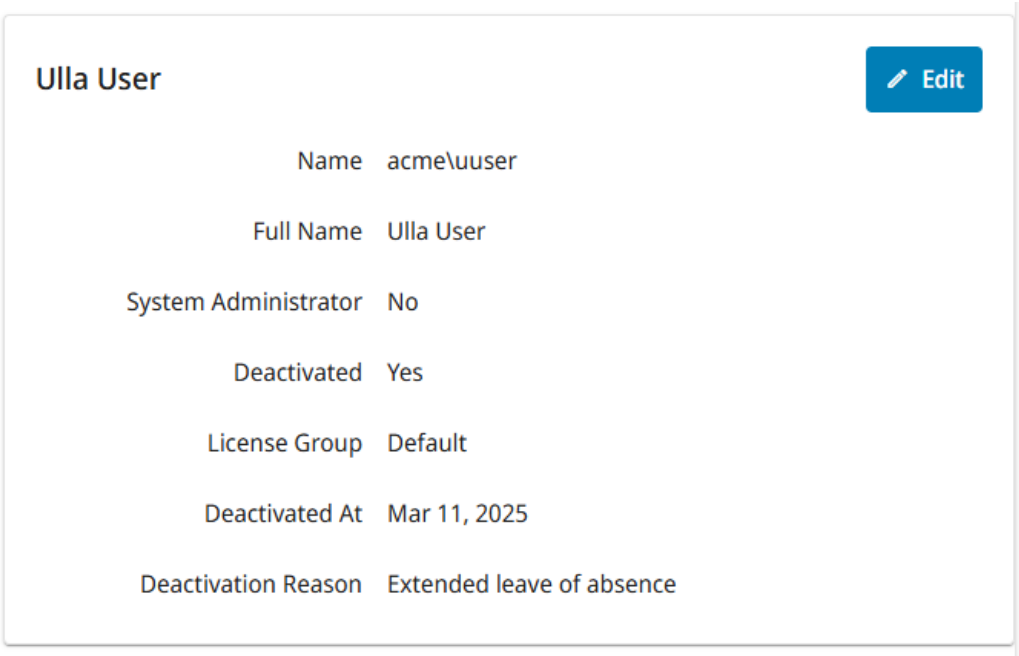
- You are a system administrator.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Users**. The User management view opens.
3. On the list of users, click the edit button for the user to edit. The Edit user details view opens.
4. Set **Deactivated** to **Yes**. The **Deactivation Reason** field is displayed.
5. Optionally, enter a comment about why the user is deactivated. This comment will not be visible to the deactivated user.
6. Click **Save**.

Results

The specified user is deactivated. The User Details view displays the time of deactivation and the deactivation reason if you entered one.



8.3.5. Deleting a user

System administrator can delete a user who should not have access to Cadmatic eShare anymore. Deleting a user also removes the user from the user list of any project or user group that the user has been assigned to and releases the user license. You cannot delete your own user account.

Prerequisites

- You are a system administrator.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Users**. The user management view opens.
3. On the list of users, locate the one to remove, and then click the delete button of that row.

   You are prompted to confirm the action.

Results

The user is removed from all projects and user groups and deleted from the system.

Related topics

User management view

8.4. Groups

In **System Admin > Groups** a system administrator can create and delete user groups, and assign users to groups. This allows project administrators to apply user permissions at data source level. That is, when defining a data source for an adapter as described in Adapters and data sources, a project administrator can select whether all users or just specific user groups are allowed to see the data obtained from that data source. Accordingly, users that are assigned to specific groups might or might not see the following entities when obtained from a data source affected by group permissions:

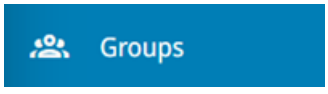
- Object attributes
- Hierarchies and visual styles
- Documents
- Indexed documents
- Search results

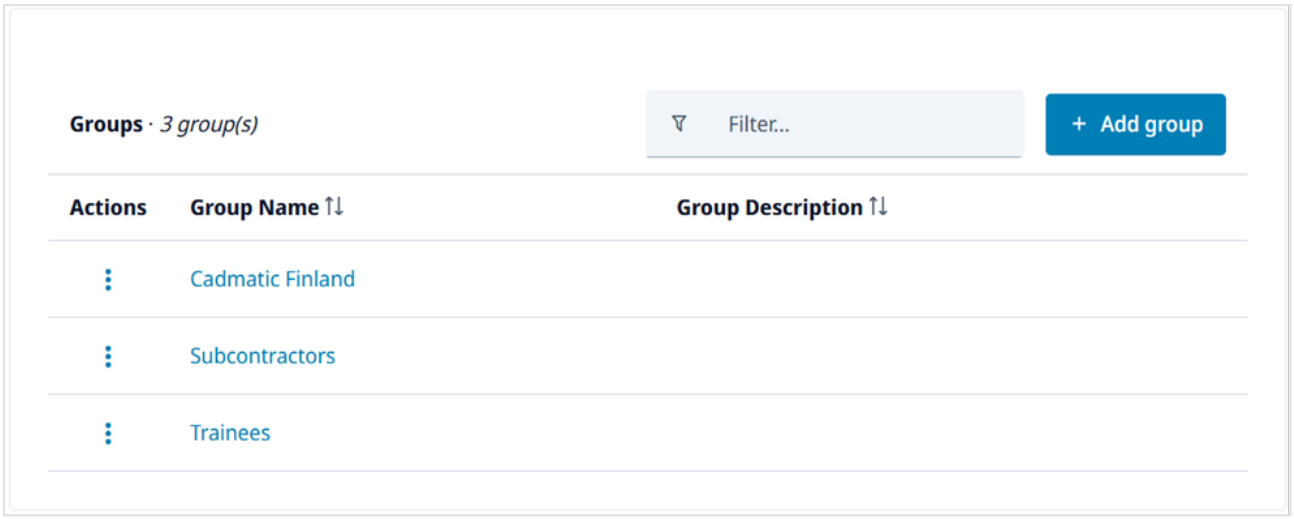
8.4.1. Group management view **266**

8.4.1. Group management view

In the Group Management view of Cadmatic eShare a system administrator can create user groups and assign users to groups.

- To open the Group Management view, select **Groups** in the menu.





The title row of the Groups list indicates how many groups there are in total.

8.4.1.1. Adding a new group

A system administrator can add new user groups to Cadmatic eShare to enable project administrators to limit users' access to specific data.

Prerequisites

- You are a system administrator.

Do the following:

1. Select **System Admin** in the main menu. The system administration view opens.
2. Select **Groups**. The group management view opens, listing the groups currently defined in the system.
3. Select **Add group**.
4. In the Group Details section, enter the following information:
    - **Name** – Enter a name for the group.
    - **Description** (optional) – Enter a description of the group.
5. Set the following options as needed:
    - **Collision assignment allowed**– When enabled, the group and any of the users belonging to the group can be set as a collision assignee in any of the projects the group is assigned to.

      To assign the group to the project, see Adding and removing project users and groups.

- **Offline use allowed**– When enabled, the users belonging to the group can take offline any of the projects which have offline use allowed.

6. In the Users section, select the **Group Member** check box of the users to be added to the new group.
7. Select **Save**.

Results

The new group is listed in the Groups list of the Group Management view.

You can add or remove group members as described in Editing group details.

Project administrators can select the group when defining data sources to specify that the group members can access data provided by the data source.

8.4.1.2. Editing group details

A system administrator can edit group's name and description, and add or remove group members.

Prerequisites

- You are a system administrator.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Groups**. The group management view opens.
3. Click ⋮ in Actions column and select **Edit**. A view for editing the group details opens.
4. Edit the details as needed and click **Save**.

Results

The changes are applied to the group.

8.4.1.3. Deleting a group

A system administrator can delete a group that is no longer needed for limiting users' access to specific data sources. Deleting a group does not delete the related users or remove them from the project.

Prerequisites

- You are a system administrator.

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Groups**. The group management view opens.
3. Click ⋮ in Actions column and select **Delete**. You are prompted to confirm the action.

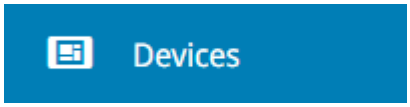
Results

The group is removed from all data source configurations and deleted from the system.

8.5. Devices

In the Devices view of Cadmatic eShare a system administrator can view, import, and export a list of eGo devices, host IDs and license information. The device list can be used to renew the license information. In the Devices view also activation keys for eGo can be added and deleted.

- To open the Devices view, select **Devices** in System Administration.



The list of devices is synchronized from eGo. Old client versions or expired licenses are highlighted in red and new licenses are highlighted in blue.

New License means that the new license has been entered in eShare but it has not yet been taken into use, because the current license in eGo is still valid.

When eGo uses activation key licensing, the time in license expiration column indicates the time of the next online verification.

8.5.1. Adding an activation key

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Devices**. The device management view opens.
3. In Activation Keys section, click **Add Activation Key**.
4. Enter the device name pattern for identifying the eGo device.

   The device name pattern can be written in either regular expression or wildcard notation, using * as a wildcard symbol. If the field is left empty, it will always match.

5. Enter the activation key in the **Activation key** field.
6. Click **Save**.

8.5.2. Editing an activation key

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Devices**. The device management view opens.
3. In Activation Keys section, click the edit button for the activation key.
4. Make needed changes.
5. Click **Save**.

8.5.3. Removing an activation key

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Devices**. The device management view opens.
3. In Activation Keys section, click the delete button for the activation key. A confirmation dialog opens.
4. Select **Remove**.

8.5.4. Renewing a license

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Devices**. The device management view opens.
3. Click the edit button for the device. A dialog with the current license key opens.
4. Enter the new license key in the field and click the save button.

Note: To renew multiple licenses, export the device list from eShare, update the license information in Excel, and import the device list to eShare.

8.5.5. Removing a device

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Devices**. The device management view opens.
3. Click the delete button for the device. A confirmation dialog with the device details opens.
4. Click **Remove**.

8.5.6. Exporting a device list to Excel

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Devices**. The device management view opens.
3. Click **Export to Excel**.
4. Open or save the list (XSLX format) to your computer.

8.5.7. Importing a device list from Excel

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Devices**. The device management view opens.
3. Click **Import from Excel**.
4. Select the Excel file (XSLX format) from your computer's file system and click **Open**. The license information of existing devices will be updated.

Note: The value in Excel column Name must match the one in eShare to update the license information for that device, and the license information in Excel column License must be in valid eGo format for the information to be updated.

Note: New devices can only be added to the list from eGo.

Click **Refresh** to update the Devices list in eShare after changes from exports or in eGo.

8.6. Licenses

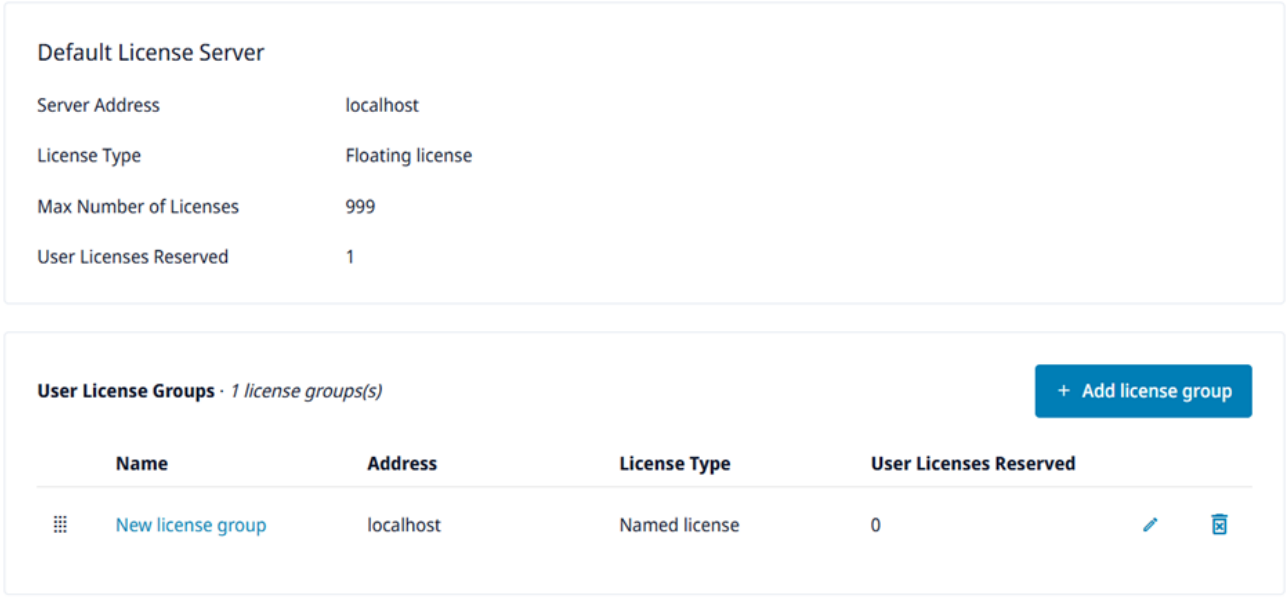
In **System Administration > Licenses** a system administrator can view details on licenses and manage them, on the application pool, see background tasks in more detail and view log messages.

- To open the Dashboard view, select **Licenses** in the menu.



Licenses are taken into use from the license server as needed when users in the license group log in. When the maximum limit of licenses is reached, either limited by the number of licenses available in the license server, or **Max number of licenses** set by the system administrator in the settings of the license group, giving a license to a user signing in is not quite as straightforward. When a user signing in needs a license, the system checks if there are licenses reserved to users who are not currently actively using eShare. The license is taken from the user with the longest inactivity and given to the user now signing in. The function of this feature is to limit the use of the licenses in a license group. It can be utilized, for example, to limit the use of licenses by a subcontractor.

The inactivity of a user is defined by either 3 minutes of not opening eShare or time of inactivity set in Internet Information Services (IIS) Manager application settings. See Configuring Application Settings.



Default License Server pane shows information on the default license server.

User License Groups pane lists user license groups. You can add, edit, or delete the groups. If the list contains more than one user license group, you can rearrange the groups by dragging. To edit an existing user license group, click on the group name to view its information.

8.6.1. Adding a new user license group

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Licenses**. The license management view opens.
3. In the **User License Groups** pane, click **Add license group**.
4. In the **Group Details** pane, specify the following:
    - **Group Name** – Enter a name for the user license group.
    - **License Server** – Enter the address of the license server.
    - **License Type** – Select if the licenses are **Named** or **Floating**.
    - **Limit Number of Licenses** – Specify if the number of licenses are **Unlimited** or **Limited**. If set to **Limited**, specify the maximum number of licenses in **Max number of Licenses** field.
5. In the **User Groups** pane, select which groups are added as group members to the user license group.
6. Select **Save**.

8.6.2. Editing license group details

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Licenses**. The license management view opens.
3. In the **User License Groups** pane, select the edit button ✎ for the license group.
4. A view for editing the license group details opens.
5. Make the changes and select **Save**.

8.6.3. Deleting a user license group

Do the following:

1. Click **System Admin** in the main menu. The system administration view opens.
2. Click **Licenses**. The license management view opens.
3. In the **User License Groups** pane, locate the user license group to delete, and then click the delete button of that row. You are prompted to confirm the action.

8.7. JSON Web Tokens

In **System Administration > JSON Web Tokens** a system administrator can generate a JSON Web Token (JWT), which is required for authenticating between design applications when eShare is using Azure AD or OIDC authentication.

Starting from 2024T3, JSON Web Tokens created in eShare conform to the standard RFC 7519 method. Due to these changes, all previously generated JSON Web Tokens will cease to be valid. After upgrading from 2024T2 or older, all JSON Web Tokens need to be generated again to continue using any integrations using JSON Web Tokens.

- To open the JSON Web Tokens view, select **JSON Web Tokens** in the menu.



8.7.1. Generating a JSON Web Token

Do the following:

1. Select **System Admin** in the main menu. The system administration view opens.
2. Select **JSON Web Tokens** in the menu.
3. Select the correct user from the **User** drop-down menu. The client will be authenticated with the selected user.
4. Select **Generate Token**. The generated JSON Web Token opens in a Generated JSON Web Token (JWT) dialog.
5. Copy the JSON Web Token and use it in the Authorization header of your service requests. After copying the Web Token, select **OK** to close the dialog.

8.8. External services

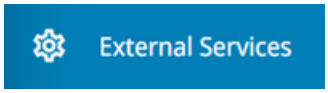
In **System Administration > External Services** a system administrator can configure AI support and map support to enable external services to be used in eShare projects.

8.8.1. AI support

In **System Administration > External Services > AI Support** a system administrator can configure if AI label detection can be used for point clouds. See AI label detection for point clouds.

For more information on Azure OpenAI, see https://learn.microsoft.com/en-us/azure/ai-services/openai/chatgpt-quickstart?tabs=command-line%2Cpython&pivots=rest-api.

- To open the AI Support management view, select **External Services** in the menu.



- Expand **AI Support** pane.

8.8.1.1. Enabling AI label detection

Prerequisites

For Azure AI:

- An Azure subscription
- An Azure Document Intelligence resource

Do the following:

1. Select **System Admin** in the main menu. The system administration view opens.
2. Select **External Services** in the menu.
3. Expand **AI Support** pane.
4. Select **Edit** in the AI Support section.
5. In AI label detection configuration section, specify the following:
    - **AI label detection**– To enable AI label detection, set **Enabled**.
6. In General section, specify the following:
    - **AI Service**– Select the AI service from the drop-down menu:
        - **Azure AI** – If you are using **Azure AI** service, specify the following:
            - **API Key** – Enter the API key for the Azure AI service.

              The API key can be found in the Keys and Endpoint section when examining the Document Intelligence resource from the Azure portal. Either KEY 1 or KEY 2 can be used.

      In Options section, specify the following:

- **Endpoint**– Specify the used endpoint.
  The endpoint can be found in Keys and Endpoint section when examining the Document Intelligence resource from the Azure portal. Example endpoint: https://docs-test-001.cognitiveservices.azure.com/

7. Select **Save**.

8.8.1.2. Disabling AI label detection

Do the following:

1. Select **System Admin** in the main menu. The system administration view opens.
2. Select **External Services** in the menu.
3. Expand **AI Support** pane.
4. Select **Edit** in the AI Support section.
5. In AI label detection configuration section, set **AI label detection** as **Disabled**.
6. Select **Save**.

Related topics

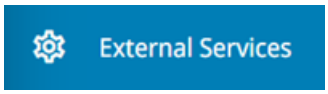
AI Assistant

AI label detection for point clouds

8.8.2. Map support

In **System Administration > External Services > Map Support** a system administrator can configure if map support is enabled or disabled. When map support is enabled, project maps can be used in eShare.

- To open the Map Support management view, select **External Services** in the menu.



- Expand **Map Support** pane.

8.8.2.1. Enabling map support

Prerequisites

For Google Maps:

- A Google Maps API key.

  If you do not have one, create a new project in Google Cloud Console: https://console.cloud.google.com/projectselector2/google/maps-apis/overview?pli=1 and the API key will be prompted, or available in the credentials page.

For Open Street Map:

- An URL for the tile provider.

  Follow the usage rules of the selected provider. List of providers is available in Open Street Map wiki: https://wiki.openstreetmap.org/wiki/Raster_tile_providers.

Do the following:

1. Select **System Admin** in the main menu. The system administration view opens.
2. Select **External Services** in the menu.
3. Expand **Map Support** pane.
4. Select **Edit** in the Map Support section.
5. In map support configuration, specify the following:

   Status – To enable map support, set to **Enabled**.
6. In General section, specify the following:
   - **Map Service** – Select the used map service from the drop-down menu:
     - **Google Maps** – If you are using Google Maps service, specify the following:
       - **API key** – Enter the Google Maps API key.
     - **Open Street Map** – If you are using Open Street Map service, specify the following:
       - **URL** – Enter the URL providing the tiles for the map. An example URL is *https://url.org/{z}/{x}/{y}.png*
7. Select **Save** at the top of the configuration view.

8.8.2.2. Disabling map support

Do the following:

1. Select **System Admin** in the main menu. The system administration view opens.
2. Select **External Services** in the menu.
3. Expand **Map Support** pane.
4. Select **Edit** in the Map Support section.

5. In map support configuration, specify the following:

   Status – To disable map support, set to **Disabled**.

9. Command-line interface for administrative tasks

You can use the eShareAdmin.exe tool of the eShare server to perform various administrative tasks from the command line. Typically the tool is located in *\Program Files\Cadmatic\eShareAdminTools\eShareAdmin*.

Run the tool without any options or parameters to display usage instructions:

```
eShareAdmin
```

Run the tool with the required option and parameters to perform a task:

```
eShareAdmin <option> <parameters>
```

The tool supports the following options and parameters.

Option	Parameters	Description
-p	uri	Lists the names of all eShare projects. *uri* – Specify the URI of the eShare server.
-u	uri projectname filename [modelImporterId] [--uploadOnly]	Uploads a model from the given filename file to the specified eShare project. *projectname* – Specify the project to which to add the model. *filename* – Specify the path and filename of the model to upload. *modelImporterId* – Specify the model importer to use. If not specified, the default EBM importer is used. *--uploadOnly* – Will not perform the publish, just uploads the model.
-g	[-c] uri groupname [projectname]	Imports users from a domain or machine group specified in *groupname*.

Option	Parameters	Description
		-c – (Optional) Use this parameter to create an eShare user group that has the same name as the imported group, and to add the imported users to this eShare user group. If the users already exist in eShare, they are added to this group. If the group already exists, an error message is displayed. If you specify a project in the *projectname* parameter, the user group is added as a user to that project. *groupname* – Specify the domain name or computer name and the name of the group from which to import users: "DOMAIN_NAME\Group Name" "COMPUTER_NAME\Group Name" *projectname* – (Optional) Specify the project to which to add the users or (if using the *-c* parameter) the user group. Do not use the parameters *-c* and *-s* at the same time.
	[-s] uri groupname [projectname]	Synchronizes Active Directory (AD) groups to eShare groups. *-s* – (Optional) Use this parameter to add users from an AD group to eShare. The parameter creates an eShare group if one does not exist yet, and adds or removes users from the eShare group to match the AD group if the group with the same name exists. Using the parameter does not remove user accounts from eShare. *groupname* – Specify the domain name or computer name and the name of the group from which to import users: "DOMAIN_NAME\Group Name" "COMPUTER_NAME\Group Name" Do not use the parameters *-c* and *-s* at the same time.
	[-c|-s] uri groupname [projectname] [--entraId] [-a] [-t]	Imports users or groups from Entra ID/Azure AD. *-c* – (Optional) Use this parameter to create an eShare group with the same name as the Entra ID group.

Option	Parameters	Description
		-s – (Optional) Use this parameter to synchronize users with and existing eShare group or creates a new group if it does not exist. *groupname* – Specify the name of the Entra ID group to import. *projectname* – (Optional) Specify the project to which to add the users or (if using the *-c* parameter) the user group. *--entraId* – (Optional) Use this parameter to specify that the group is imported from Entra ID. *-a* – (Optional) Specify the path to a JSON file with Entra ID authentication credentials. See Auth JSON file format. *-t* – (Optional) Supplies the JSON Web Token. Do not use the parameters *-c* and *-s* at the same time.
-t	[-t]	Supplies the JSON Web Token for Azure Active Directory (AD) that is defined in system administration user interface in eShare.

Examples

To run a command, you need to know the URI of the Cadmatic eShare server. In the examples below the server address and port is *localhost:81*.

- List all projects in a given eShare server:
  `eShareAdmin -p http://localhost:81`
- Update the 3D model of a project by uploading a model file:
  `eShareAdmin -u http://localhost:81 MyProject MyModel.ebm`
- Import users from a domain group:
  `eShareAdmin -g http://localhost:81 "My_Domain\My Group"`

- Import users from a domain group to an eShare user group by the same name, and add the user group (not the individual users) to a project:
  `eShareAdmin -g -c http://localhost:81 "My_Domain\My Group" MyProject`
- Synchronizes AD groups to eShare groups:
  `eShareAdmin -g -s http://localhost:81 "MyDomain\My Group" MyProject`
- Supplies the Azure Active Directory (AD) token:
  `eShareAdmin -p http://localhost:81 -t <token>`

9.1. Auth JSON file format

```
{
  "tenantId": "YOUR_TENANT_ID",
  "clientId": "YOUR_CLIENT_ID",
  "clientSecret": "YOUR_CLIENT_SECRET"
}
```

The credentials file can be either created manually or by generating it with the new admin tool command *generateAuthTemplate <outputPath>*.

For example:

`eShareAdmin.exe generateAuthTemplate C:/temp/auth.json`

will generate a JSON file with the correct format and placeholder values into the given output path.

10. eShare Backup Restore tool

You can backup and restore Cadmatic eShare server, server and project databases, server configuration files, and individual eShare projects using eShare Backup Restore tool from the command line by running it as an administrator. The *eShareBackupRestore.exe* is located in *Program Files\Cadmatic\eShareAdminTools\eShareBackupRestore*.

Any files referenced by data sources are not owned by eShare and they are *not* included in the backups.

Note: Running the tool stops eShare server and restarts it again after the tool has finished running.

Prerequisites

- Cadmatic eShare server must have appropriate permissions in the database. It should be able to create, delete, and rename databases, and also be able to insert, update, delete, and select data in the database tables. See Required permissions for eShare Backup Restore tool.

Do the following:

Run the tool from the command line without any options to display usage information.

```
eShareBackupRestore
```

10.1. Project backup

Perform the following to create a backup of Cadmatic eShare project.

```
eShareBackupRestore backupproject <option>
```

Required options	Description
-u --url	eShare server's URL
-d --directory	directory where to store the backup file
-p --project	name of the project to backup
Optional options	**Description**
--pointclouds	flag which specifies whether to backup point clouds and textured meshes
--force	flag which specifies whether to not ask for confirmation before proceeding with the backup
--loglevel	sets the logging level: Trace, Debug, Information, Warning, Error, or Critical. Default level is Information.
--help	show help and usage information

Examples

To run a command, you need to know the URI of the Cadmatic eShare server. In the examples below the server address and port is *localhost:81*.

- Creates a backup of the project:
  ```
  eShareBackupRestore backupproject -u http://localhost:81 -p DemoProject -d c:\temp\
  ```
- Creates a backup of the project, including point clouds and textured meshes:
  ```
  eShareBackupRestore backupproject -u http://localhost:81 -p DemoProject -d c:\temp\ --pointclouds
  ```
- Creates a backup of the project without asking for confirmation:
  ```
  eShareBackupRestore backupproject -u http://localhost:81 -p DemoProject -d c:\temp\ --force
  ```

10.2. Server backup

Perform the following to create a backup of Cadmatic eShare server.

```
eShareBackupRestore backupserver <option>
```

Required options	Description
-u --url	eShare server's URL
-d --directory	directory where to store the backup file
Optional options	**Description**
--pointclouds	flag which specifies whether to backup point clouds and textured meshes
--force	flag which specifies whether to not ask for confirmation before proceeding with the backup
--loglevel	sets the logging level: Trace, Debug, Information, Warning, Error, or Critical.

Required options	Description
	Default level is Information.
--help	show help and usage instructions

Examples

To run a command, you need to know the URI of the Cadmatic eShare server. In the examples below the server address and port is *localhost:81*.

- Creates a backup of the server:
  `eShareBackupRestore backupserver -u http://localhost:81 -d c:\temp\`
- Creates a backup of the server, including point clouds and textured meshes:
  `eShareBackupRestore backupserver -u http://localhost:81 -d c:\temp\ --pointclouds`
- Creates a backup of the server without asking for confirmation:
  `eShareBackupRestore backupserver -u http://localhost:81 -d c:\temp\ --force`

10.3. Restoring eShare project from backup

Perform the following to restore a Cadmatic eShare project from backup.

```
eShareBackupRestore restoreproject <option>
```

Required options	Description
-u --url	eShare server's URL
-f --file	the backup file to restore from
-p --project	Original name of the project to restore if the backup file contains multiple projects. If the file contains only one project this is not needed.

Required options	Description
Optional options	**Description**
--newdb	renames the project database If another database with the name already exists, it will not be overwritten
--force	flag which specifies whether to not ask for confirmation before proceeding with the restore
-c --connectionstring	can be used to override the IIS connection string
--loglevel	sets the logging level: Trace, Debug, Information, Warning, Error, or Critical. Default level is Information.
--map-users	maps each user the backup refers to onto the target server account with the same login, when the account IDs differ. Use this when moving a project between servers that have the same users under different account IDs. See Mapping users.
--user-map-file <path>	maps users by the pairs listed in a JSON file. Use it when logins differ between the servers. Used alone, it maps only the pairs in the file. Add --map-users to match the rest by login. See Mapping users.
--help	show help and usage information

10.3.1. Mapping users

Each key in the mapping file is a user from the backup, written as a login or an account ID. Each value is a login on the target server. Example of the mapping file:

```
{
"jdoe@example.com": "john.doe",
"3f2504e0-4f89-11d3-9a0c-0305e82c3301": "jane.doe"
}
```

Notes:

- Logins are matched case-insensitively.
- When both options (--map-users and --user-map-file) are used, the pairs in the file win over automatic matches.
- Each target account can receive only one user. If two users from the backup map to the same account, the restore stops and lists the conflicting pairs.
- A backup taken before 2026H2 only knows the users who had a permission on the project. Server administrators and users who got access through a group can only be mapped by account ID in the mapping file. The restore warns about this. Taking a new backup on an up-to-date server removes the limitation.

Examples

To run a command, you need to know the URI of the Cadmatic eShare server. In the examples below the server address and port is *localhost:81*.

- Restores the project from backup, when there is only one project in the backup file:
  ```
  eShareBackupRestore restoreproject -u http://localhost:81 -f c:\temp\eShareServer_DemoProject_2024T1R4_2024-04-04-123456.7z
  ```
- Restores the project from backup when there are multiple projects in the backup file:
  ```
  eShareBackupRestore restoreproject -u http://localhost:81 -f c:\temp\EShareProject_DemoProject_2024T1R4_2024-04-04-123456.7z -p DemoProject
  ```
- Restores the project from backup and renames the project database with a new name:
  ```
  eShareBackupRestore restoreproject -u http://localhost:81 -f c:\temp\EShareProject_DemoProject_2024T1R4_2024-04-04-123456.7z --newdb DemoProjectDatabase
  ```
- Restores the project from backup and overrides the IIS connection string:
  ```
  eShareBackupRestore.exe restoreproject -u http://localhost:81 -f c:\temp\EShareProject_DemoProject_2024T1R4_2024-04-04-123456.7z -c "server=localhost;initial catalog=EShare; uid=sa; pwd=PassWd123;"
  ```

10.4. Restoring eShare server from backup

Perform the following to restore Cadmatic eShare server from backup.

```
eShareBackupRestore restoreserver <option>
```

Required options	Description
-u --url	eShare server's URL
-f --file	the backup file to restore from
Optional options	**Description**
--configs	flag which specifies whether to restore the server's configuration files These files are *AppSettings.config*, *ConnectionStrings.config*, and *Web.config* in the installation directory of eShare server, and *NLog.config* in the *bin* folder of the installation directory of eShare server.
-c --connectionstring	can be used to override the IIS connection string
--force	flag which specifies whether to not ask for confirmation before proceeding with the restore
--loglevel	sets the logging level: Trace, Debug, Information, Warning, Error, or Critical. Default level is Information.
--help	show help and usage information

Examples

To run a command, you need to know the URI of the Cadmatic eShare server. In the examples below the server address and port is *localhost:81*.

- Restores the server from backup:
  `eShareBackupRestore restoreserver -u http://localhost:81 -f c:\temp\EShareServer_2024T1R4_2024-04-04-123456.7z`
- Restores the server from backup, including the server's configuration files:

```
eShareBackupRestore restoreserver -u http://localhost:81 -f
c:\temp\EShareServer_2024T1R4_2024-04-04-123456.7z --configs
```

- Restores the server from backup without asking for confirmation:
  ```
  eShareBackupRestore restoreserver -u http://localhost:81 -f
  c:\temp\EShareServer_2024T1R4_2024-04-04-123456.7z --force
  ```
- Restores the server from backup and overrides IIS connection string:
  ```
  eShareBackupRestore restoreserver -u http://localhost:81 -f
  c:\temp\EShareServer_2024T1R4_2024-04-04-123456.7z -c
  "server=localhost;initial catalog=EShare; uid=sa; pwd=PassWd123;"
  ```

10.5. Required permissions for eShare Backup Restore tool

Using eShare Backup Restore tool requires the following permissions and settings to Windows, SQL Server, and Internet Information Services (IIS) Manager.

10.5.1. SQL Server permissions

The SQL principal is whoever the eShare connection string in *ConnectionStrings.config* authenticates as:

- **Integrated Security** (Windows Auth): the principal is the Windows account running the tool (the elevated administrator).
- **SQL Auth** (SQL Authentication): the principal is the SQL login whose credentials are stored in *ConnectionStrings.config*.

The `-c / --connectionstring` flag overrides this; in that case the principal is whatever the override string authenticates as.

10.5.1.1. Per-verb SQL permissions

Verb	Server-level role	Main database (EShare or per Initial Catalog)	Project databases
`backupproject`	none required	`db_datareader`	`db_datareader` on every project database

Verb	Server-level role	Main database (EShare or per Initial Catalog)	Project databases
`backupserver`	none required	`db_datareader`	`db_datareader` on every project database
`restoreproject`	`dbcreator` (only when the target project database does not yet exist)	`db_owner` (DELETE / INSERT into `Project`, `UserAccount`, `Permission`, schema migrations)	`db_datareader` on every project database and `db_owner` on the target project database
`restoreserver`	`dbcreator` (for any project database that does not yet exist)	`db_owner`	`db_owner` on every project database referenced by the backup

Notes:

- `restoreserver --configs` overwrites *ConnectionStrings.config*. After that point the tool re-initialises and may target a **different** SQL principal than it started with. Both must satisfy the requirements above.
- The tool runs schema migrations during restore, this requires DDL rights on the target database. `db_ddladmin` alone is not sufficient because the tool also performs bulk-copy with `SET IDENTITY_INSERT` , which requires ALTER on the user table, so `db_owner` is the floor.

10.5.2. Windows / command-line prerequisites

All apply to every verb.

- Console launched with **Run as Administrator** (UAC-elevated). The *Microsoft.Web.Administration.ServerManager* API the tool uses requires elevation to read *C:\Windows\System32\inetsrv\config\applicationhost.config* and to stop/start the eShare site and application pool.
- Read access to the eShare site's physical path (typically *C:\Program Files\Cadmatic\eShare*). This is needed to read *Web.config*, *AppSettings.config*, and *ConnectionStrings.config*.

- Write + Delete access to the eShare site's physical path and to *%ProgramData%\Cadmatic\eShare* (only required by `restoreserver --configs`).
- Write access to the backup destination passed with `-d / --directory` (backup verbs) or read access to the backup file passed with `-f / --file` (restore verbs).
- Write access to *%TEMP%\eShareBackupRestore* , the tool unpacks and stages files there.
- Network reachability to the eShare URL passed with `-u / --url` . The tool issues `GET <url>/api/ServerInfo` at startup to read the server version.

10.5.2.1. eShare data folders

Backup verbs read these folders and restore verbs write them. If these have been moved to a non-default location, check *AppSettings.config* for the override key.

Folder	Default path	AppSettings.config override key
EBM (models, scenes)	*%ProgramData%\Cadmatic\eShare\ebm*	Cadmatic.EbmFolderName
Document	*%ProgramData%\Cadmatic\eShare\document*	Cadmatic.DocumentFolderName
Maps	*%ProgramData%\Cadmatic\eShare\maps*	Cadmatic.MapsFolderName
Point cloud	*%ProgramData%\Cadmatic\eShare\pointCloud*	Cadmatic.PointCloudFolderName

During server backup/restore, the tool also reads/writes two folders under the eShare site root:

- *App_Data\icons* (POI icons)
- *App_Data\markupReportTemplates* (markup report templates)

Note: During restore, the tool automatically grants *IIS APPPOOL\<eShareAppPoolName>*. Modify rights on any files it writes into these folders. This is required for eShare to read the restored files at runtime.
