



Monarch

SolidWorks to Onshape Migration

V 1.5

Created by CADSharp

www.cadsharp.com

Please email all questions and issues to keith@cadsharp.com with a subject line containing “Monarch”

User Manual

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Overview

Monarch is a SolidWorks to Onshape migration tool. Monarch performs bulk imports of both CAD and non-CAD files, re-creates assemblies, replicates folder structure, all while preventing file duplicates.

Free Trial

The free trial of Monarch is limited to 20 files per upload. This includes referenced files through either non-cad or assembly references. A full yearly license can be purchased through the Onshape app store. Purchasing a license does not require re-installing Monarch. The Monarch user needs to have the seat assigned to them in Onshape.

If you have acquired a license key, this can be entered via the top menu “License” option. The license key provided must match the email of the Onshape account being used.

Prerequisites

To use Monarch, you will need both SolidWorks on your local computer and an Onshape account. SolidWorks needs to have a valid license to launch, open files, and read file properties.

Files to be uploaded need to exist on the User’s local computer, with properties linked to the file properties in SolidWorks.

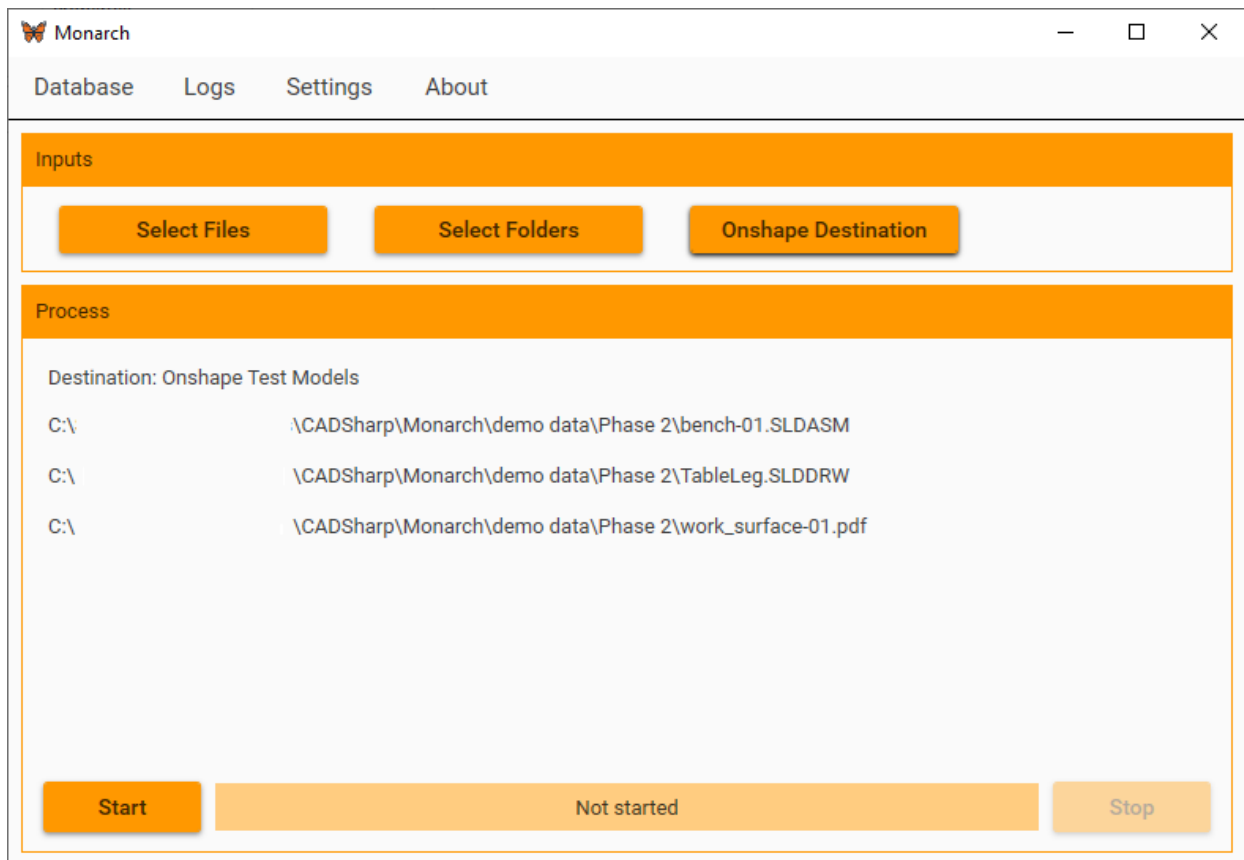
For PDM users, you may need to do “Get Latest Version” before migrating with Monarch.

Running Monarch

Included with your Monarch download is a “Monarch.exe” file. To run Monarch, simply open this file (double click).

When Monarch is run, the Onshape Login page will be opened in your default browser. The first time Monarch is run, a permissions page will be shown requesting the user to grant access to Monarch. On subsequent runs, if the user is already logged into Onshape, the login step may be skipped through automatically. Once logged in, Monarch is ready to use.

An important note on permissions: The user that logs into Onshape with Monarch does matter. Monarch acts on behalf of the user, including their Onshape permissions. Users with limited access to their enterprise may have issues using Monarch. It is recommended that Monarch is used by an Admin who can create documents seen by all users.



Setting up Monarch

The first time Monarch is run, there is some setup required. Clicking the “settings” button will open the settings window. Please fill out each of the following sections of the settings window and click **SAVE**.

Part Mappings

When uploading to Onshape, a library of property mappings needs to be created to ensure that metadata is copied from SolidWorks to Onshape.

When selecting mappings, the user needs to select an Onshape metadata property, provide the name of the SolidWorks custom property to map to, and the category.

If the ‘configuration’ category is selected, “Default to Custom” is checked, and if there is no configuration, or the specific configuration lacks that property, Monarch will instead read the “custom” property. If the property is not present in a model, it will be ignored. See also Complex Mappings

Monarch Settings

Part Mappings

Assembly Mappings

Drawing Mappings

File Uniqueness

Document Naming

Versions

Ignore Files

Misc Options

Non-CAD Files

Solidworks Drawings

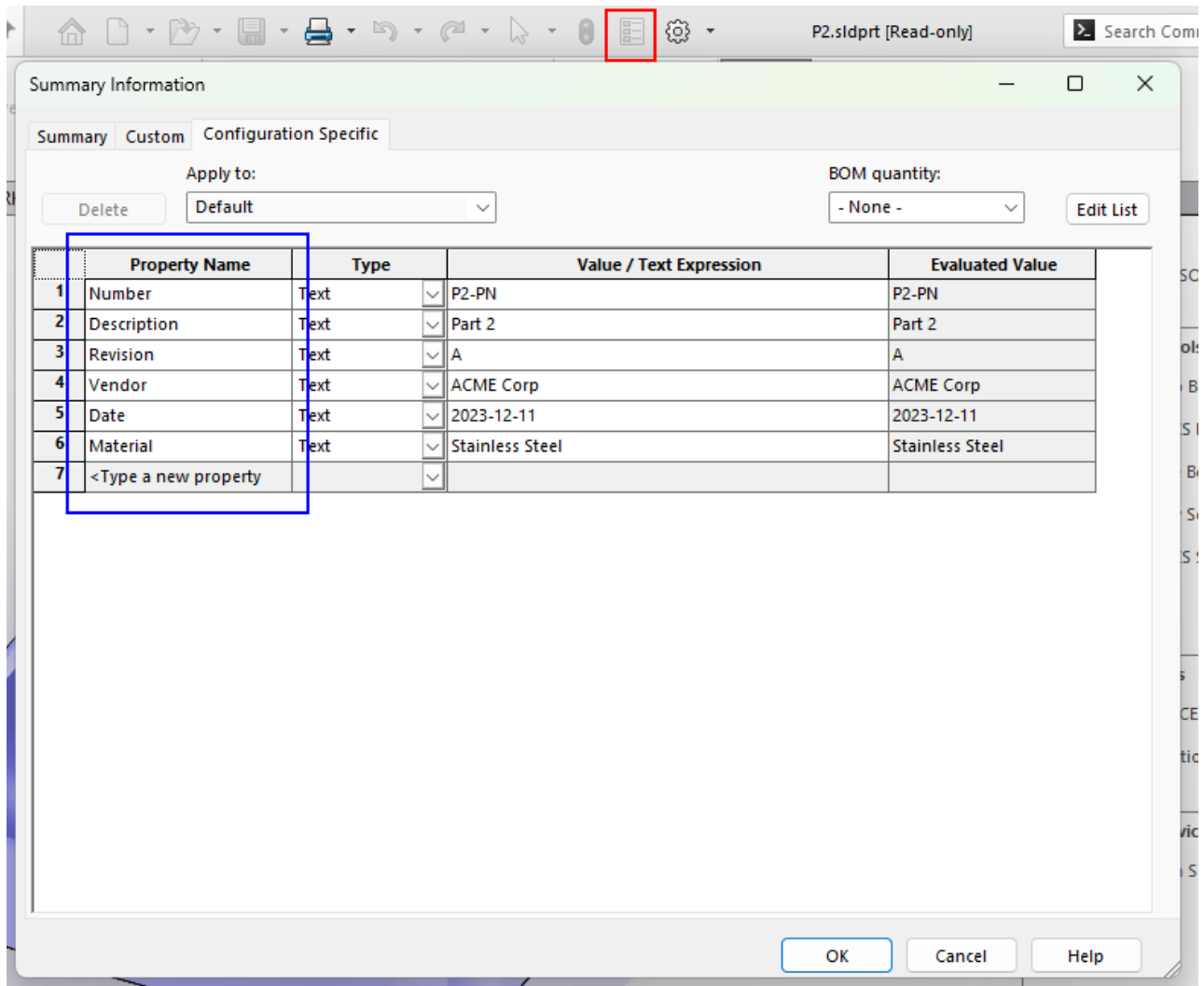
Folders

Onshape Part Property	Solidworks Property N...	Solidworks Property C...	Default to Custom
<input type="checkbox"/> Description	Description	CONFIGURATION	<input type="checkbox"/>
<input type="checkbox"/> Name	{filename}	CONFIGURATION	<input checked="" type="checkbox"/>
<input type="checkbox"/> Part number	PartNumber	CUSTOM	<input checked="" type="checkbox"/>
<input type="checkbox"/> <div></div>			<input type="checkbox"/>

Delete selected

SaveSave and Close

In the image below, the File Properties in SolidWorks are indicated by the red circled button. The area circled in blue corresponds to the “SolidWorks Property Name” that needs to be included in the Monarch settings for all Mappings.



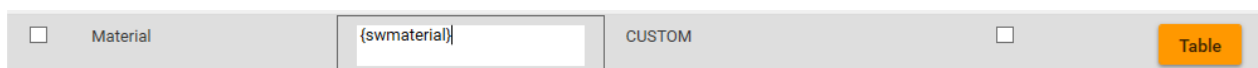
Part Materials

A specific note on part material mappings. Monarch has two ways of setting the material property in Onshape.

1. Using a custom property
2. Using the assigned material

Custom Property – In the *Custom Property* Dialog, mapping the Onshape Material from a field in the Custom Properties page in SolidWorks could result in a material in Onshape that has the name set to the value from SolidWorks, while the density is kept at 0. This is because the dialog allows text values to be specified.

SolidWorks Assigned Material – The material set in SolidWorks in the feature tree can also be read. To set this to be the value, the SolidWorks Field Name should be set to {swmaterial}. In this case, the density in Onshape will match the density in SolidWorks.



If the material name is mapped with the above steps, the material density can also be mapped by using the “Material Density” metadata property from the Onshape list. To specify units, see the “Miscellaneous Options” section for the density units’ option.

Drawing and Assembly Mappings

Drawing and Assembly mappings work the same as part mappings but are applied to assemblies in Onshape. A separate list of mappings needs to be maintained, even if there is overlap between parts and assemblies.

Complex Mappings

The source data for a field mapping can utilize data that is not in the custom properties in SolidWorks.

The following special fields are available, and *are case sensitive*:

{filename}
{configuration}
{title}
{subject}
{savedate}
{comment}
{keywords}
{author}
{swmaterial}

{sw:PROPERTYNAME}, where *PROPERTYNAME* may be replaced with any custom property name.

Filename will become the file name without the extension,

Configuration will become the name of the current configuration and is only available for configuration specific properties.

Custom properties can be combined using the wrapper {sw:propertyname}. For example, I could use description {sw:description}, and the entire wrapped property will be replaced with the value of that property.

See the table below for field combination examples:

Property combination	Example result
{sw:partno}_{filename}	52001_Flange01
{sw:partnumber} R{sw:rev}	52001 R02

Where all characters outside of the property names will be included. Such as the underscore in the first example, and the “R” in the second example.

Complex field mappings are supported everywhere field mappings are, such as File Uniqueness, Document naming, and Ignoring files.

Mapping Tables

Property mappings can have a translation table. The translation table provides a way to translate existing SolidWorks custom property values to new values in Onshape. To achieve this, specify a Table option. When the value in SolidWorks is found in the table, the value is replaced with the provided “Onshape Value”.

Values that are not found in the table are kept as is.

For example:

Value found in SolidWorks (SolidWorks Value)	...Becomes in Onshape (Onshape Value)
Approved by Engineering	Approved
Preproduction state	Preproduction
Finished	Production

Setting default values

A specific default value can be set for properties by using the table mapping.

By setting the SolidWorks value to “” (a blank value), and including a value in the Onshape value, any missing values will default to the provided value.

This can be used with a complex mapping for the SolidWorks property name set to {sw:null} to ensure that every part/assembly uploaded will have the default value set.

Category Mapping

To map an Onshape Category for parts/assemblies/drawings, first start by selecting the Onshape property type “Category”. Next, click the “table” button for the row. A different property table window will appear where the Onshape property values are Categories in your Onshape. Select a category and provide the corresponding SolidWorks value to look for.

At this time, only one Category can be mapped to a part/assembly.

File Uniqueness

Monarch is able to upload parts and assemblies to Onshape without creating duplicates by using uniqueness rules and creating a database. Two options are available to a user to define what makes a part “unique”. The options are the file path, or a SolidWorks property.

If the file path is chosen, a part will be identified by its local file path.

If a property is given, such as part number, the property will define the uniqueness of the part.

Once uploaded to Onshape, Monarch will save a database mapping the uniqueness property of a part/assembly to the specific element/document/version within Onshape.

The version of the part also defined the uniqueness, see the “Versions” section for more information.

Example:

While using a Part Number for uniqueness, I have uploaded a part:

../Flange01.sldprt – with property **PartNumber** and value 54001

I proceed to upload an assembly containing a copy of the part:

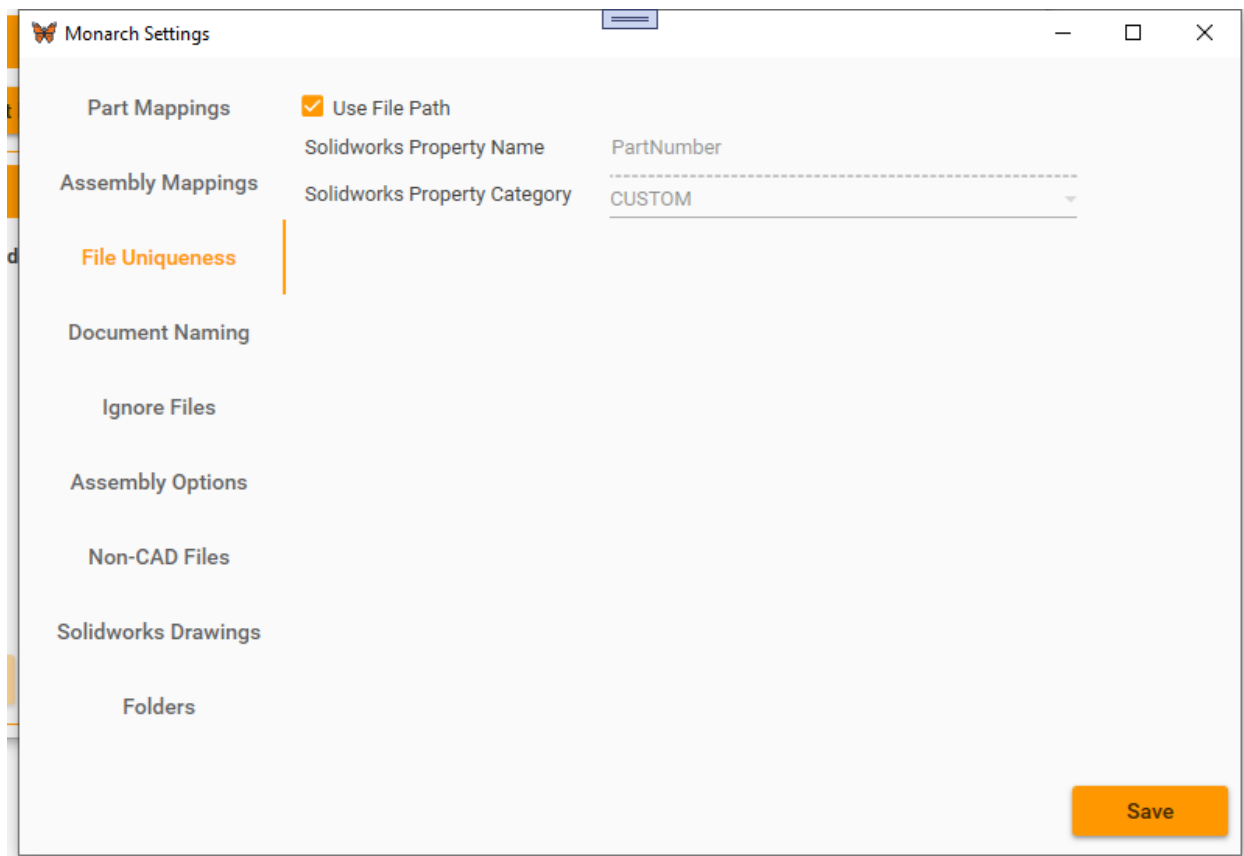
../Flange01 – Copy.sldprt – with property **PartNumber** and value 54001

When the assembly is recreated in Onshape, the copy of the flange will be checked (as it is the referenced part in SolidWorks), and the PartNumber of 54001 will be read. Because I already have a part with part number 54001 in my Onshape (Flange01.sldprt), the existing part will be inserted into my Onshape Assembly. If the file path had been selected as the uniqueness property, both the original and the copy of the Flange part would be uploaded to Onshape, with the copy inserted into my example assembly.

Setting a uniqueness property is recommended over the file path, as it will catch and fix cases like the example provided where a SolidWorks user has made a duplicate file referencing an existing part number.

The uniqueness mapping allows for complex maps, the same as all other mappings. For example, the uniqueness of a file could be both the part number and revision by using:

{sw:partnumber}{sw:rev}



Document Naming

The name of a document cannot be set in the Part or Assembly field mappings.

The options for the created document names are either the filename (without extension) or a field mapping. The field mapping must exist in both assembly and part files. If a field mapping is chosen, but the field is not available, Monarch will default to the filename.

Document naming supports complex field mappings.

Versions, Version History, and Releases

When documents are created in Onshape by Monarch, a version is created. This is a requirement for building referenced assemblies (as assemblies must reference a version for parts imported from other documents). Monarch has a few options on how to handle versions and releases.

Version History

Monarch is able to create "version history" where each version of a part or assembly in Onshape is replaced with a modified file. This is equivalent to using the "update file" function in Onshape for an imported part. For assemblies, the entire assembly is deleted and replaced with the updated version.

It is important to note that creating version history is meant as an automated process. Updating in progress or documents modified in Onshape by a user is not recommended as any progress a user has made will be removed. This includes changes to the part studio, metadata, or assembly features.

Version history does not need to be enabled, simply upload an existing part with a new version value, and the history will be created.

Version history cannot go backwards, an older version will not be uploaded if a new version exists. For example if versions B and C have been uploaded, attempting to upload version A in a subsequent upload will result in Monarch ignoring the version A file.

Option 1 – Set Default value

By selecting the checkbox for *Use Default Value* and including a value (example seen below of “V1”), all versions created by Monarch will be named with this value.

Version history can be created by changing the default version value to a “higher” value (numerically or alphabetically higher), and then uploading the files again.

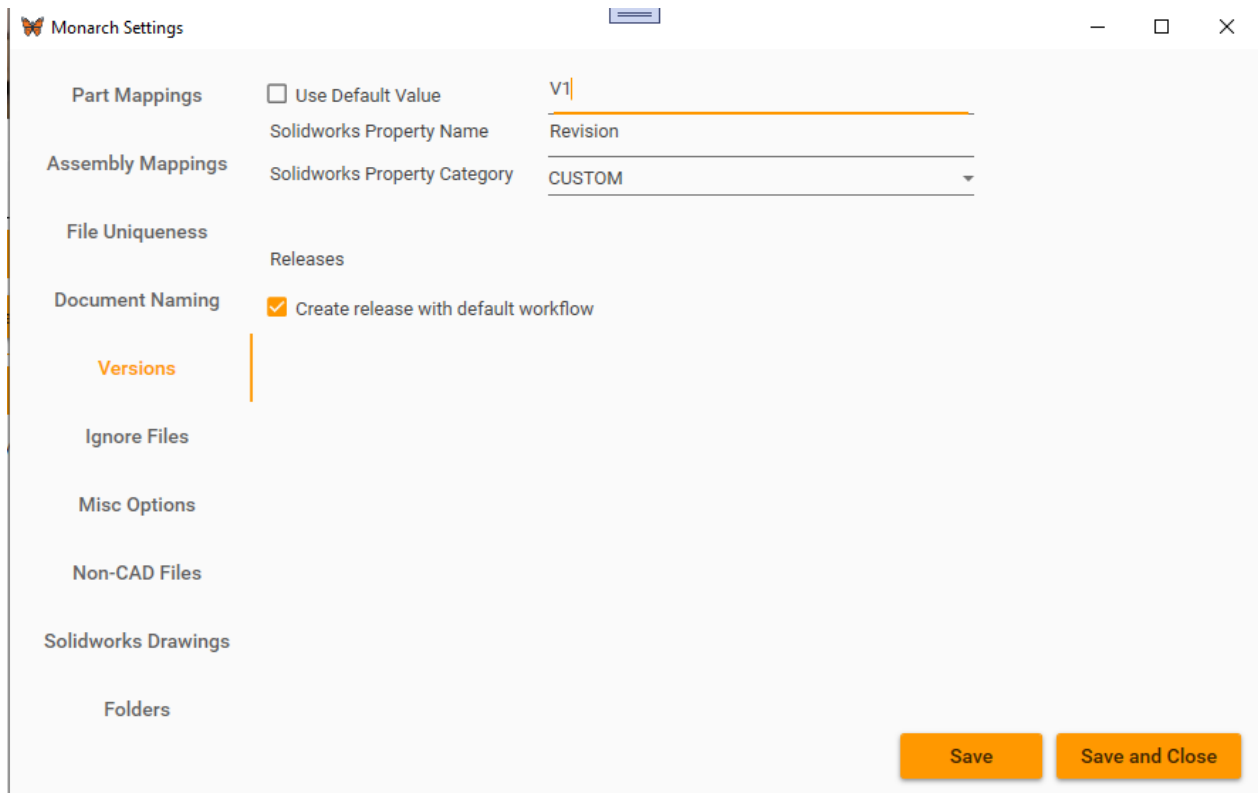
Option 2 – Map the version name

By mapping the version name to a property in SolidWorks, multiple versions of parts and assemblies can be uploaded at the same time. If these files are in separate folders (or have a mapped property for uniqueness), Monarch will order the versions in ascending order. Generally it is recommended to use the Revision or Version of the part as the name of the version in Onshape.

Releases

After creating a version, Monarch is able to release parts and assemblies. Releases are done with the default Onshape workflow. Each release is done with the individual part or assembly, and no approvers or watchers are added. All parts and assemblies are moved straight to the release state.

It is important to note that Monarch will not create part numbers. The part number field needs to be mapped for both assemblies and parts for releases to be created. Pay close attention to configured parts, each configuration needs to have a unique part number, otherwise only one of the configurations will be released.

The image shows a 'Monarch Settings' dialog box with a sidebar on the left and a main configuration area on the right. The sidebar contains the following items: 'Part Mappings', 'Assembly Mappings', 'File Uniqueness', 'Document Naming', 'Versions' (highlighted with an orange bar), 'Ignore Files', 'Misc Options', 'Non-CAD Files', 'Solidworks Drawings', and 'Folders'. The main area is currently displaying the 'Part Mappings' settings. It includes a checkbox 'Use Default Value' which is unchecked. Below it are two input fields: 'Solidworks Property Name' with the value 'Revision' and 'Solidworks Property Category' with a dropdown menu set to 'CUSTOM'. At the top of the main area, there is a text input field containing 'V1'. At the bottom right of the dialog, there are two orange buttons: 'Save' and 'Save and Close'.

Monarch Settings

Part Mappings

☐ Use Default Value

Solidworks Property Name: V1

Solidworks Property Category: Revision

Assembly Mappings

Solidworks Property Category: CUSTOM

File Uniqueness

Releases

Document Naming

☒ Create release with default workflow

Versions

Ignore Files

Misc Options

Non-CAD Files

Solidworks Drawings

Folders

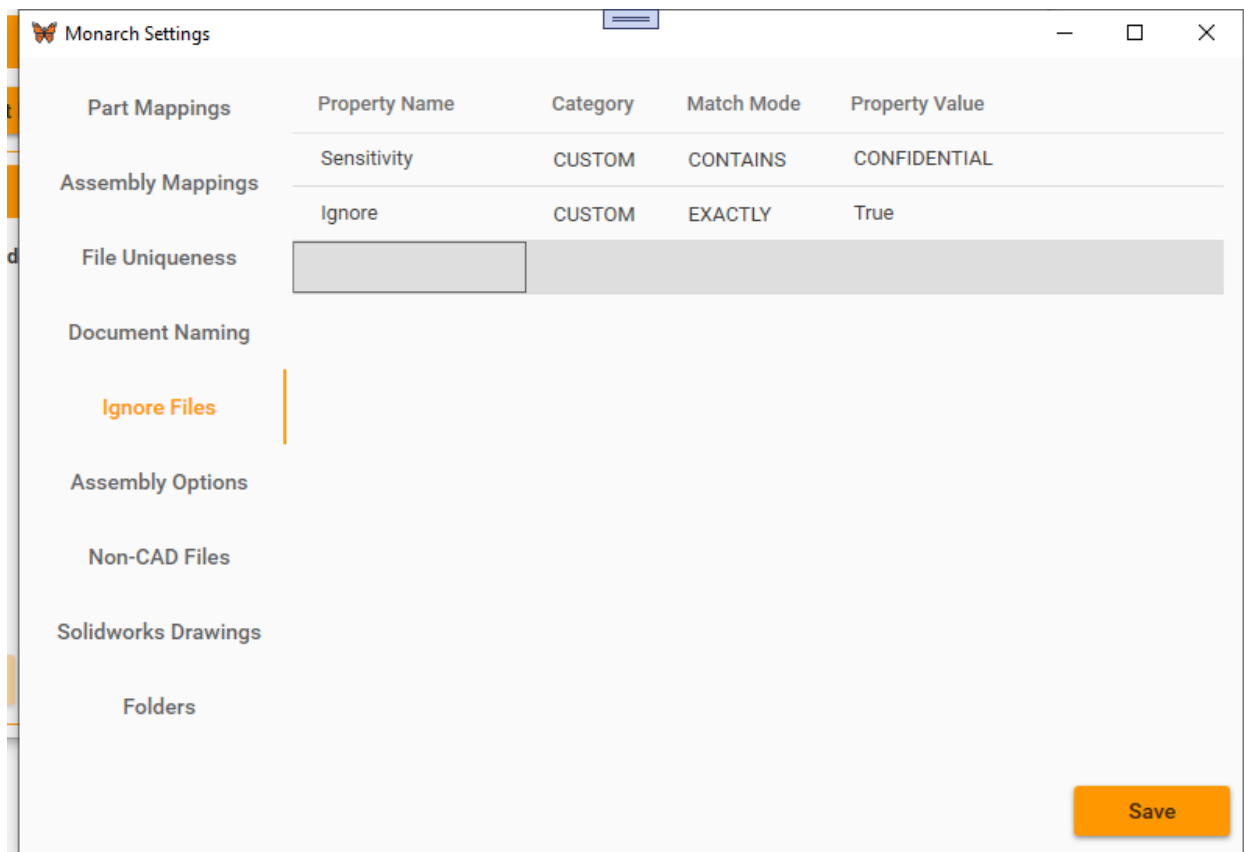
Save Save and Close

Ignore Files

It is possible that not all files need to be uploaded to Onshape. These could be toolbox, purchased, or obsolete parts. Monarch provides an option to ignore parts and assemblies.

The Ignore Files section utilizes field mappings (supporting complex field mappings), along with two value check modes: exactly and contains. Exactly will check that the value of the mapping matches exactly, this includes case sensitivity. Contains will check that the provided value is anywhere inside of the mapped value.

Files are ignored if they match any of the criteria in the list.



Miscellaneous Options

For rebuilt assemblies, Monarch gives the option to group all assembly items together as they are when the assembly was created.

Mirroring the option in Onshape's import, models can be imported with "Import orientation Y-Axis Up"

If the destination folder in Onshape is always the same, the default folder can be set by selecting it in the Misc options. When Monarch starts up, the specified folder will be set as the destination.

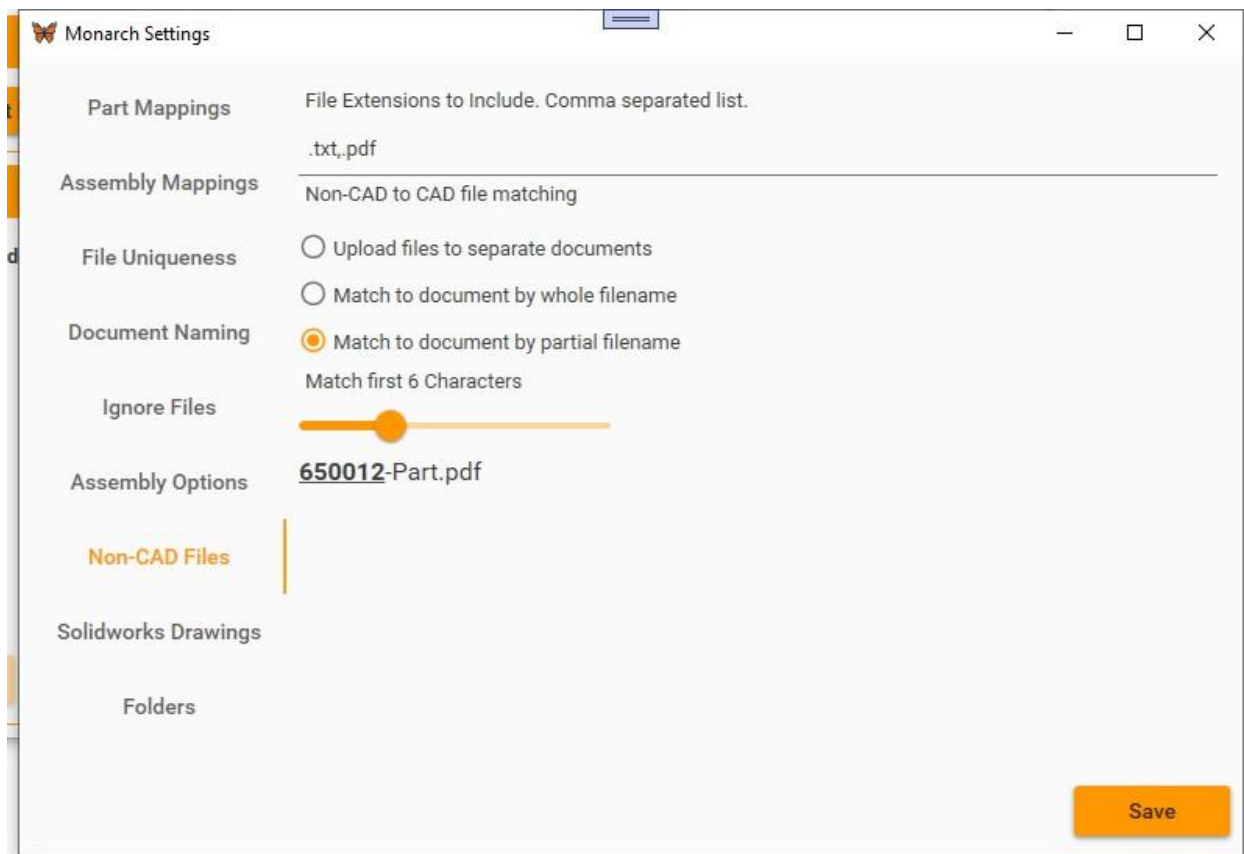
Non-CAD Files

Non-CAD Files can be included either explicitly by including them in the file selection, or automatically with selected folders. For automatic inclusion, the file extensions need to be added to the settings page.

The list of extensions is a comma separated list, for an example please see the image below.

Non-CAD files can be uploaded to their own document, or associated by filename.

If associating by filename, the options are either by entire filename (minus the extension), or by the first X characters. If the first X characters is selected, the filename will be matched to a maximum of either the selected number or the length of the filename. If a file is unable to be associated to a document, it will be uploaded to its own document.



SolidWorks Drawings

The following section applies to specifically “.slddrw” file types. PDF, DXF, DWG, etc. file types are covered by the Non-CAD file section.

When “Export Drawings as PDF” or “Export Drawings as DXF” is selected, Monarch will search selected source folders for SolidWorks drawing files.

If the Upload to new document is selected, the drawing will be exported to its own document in the Onshape destination.

If the option for associating the drawing is selected, the exported PDF/DXF will be uploaded to the same document as the part referenced in the drawing.

Folders

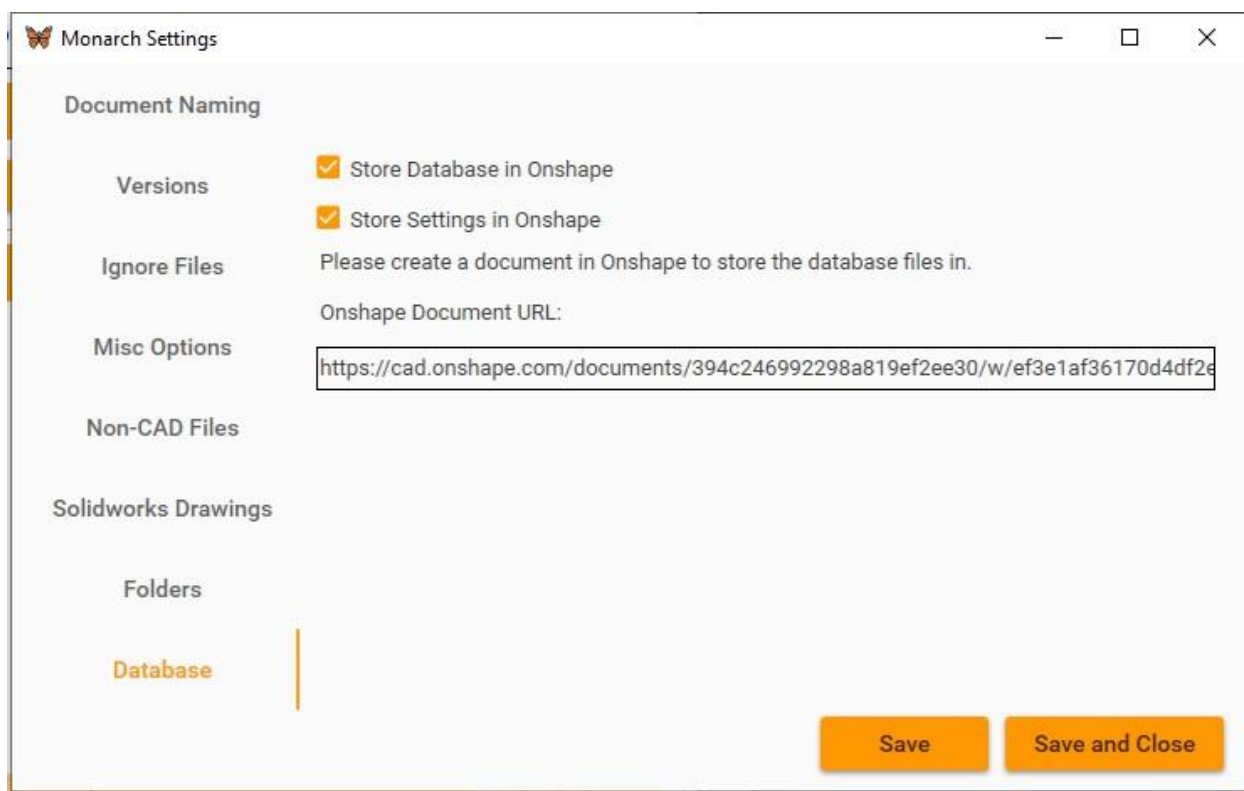
When selecting a source folder for upload, Monarch is able to replicate sub folder structure. Please see the folder section of Uploading with Monarch for more details.

Database

As of V1.4, Monarch can now store the database and settings in Onshape. The database and settings files are also stored locally when this option is selected, changing this setting should have no impact on using Monarch.

When the options for Store Database or Store Settings in Onshape are selected, the user must provide a URL for an Onshape document. Monarch will upload the files into this document in Onshape. When Monarch is launched, the local database and settings files will be replaced with the files in the Onshape document.

Note, that while this does allow for multiple users to share a database, the database is only downloaded and uploaded when Monarch is launched or when a full upload process is complete. Users cannot simultaneously use Monarch.



Uploading with Monarch

Once all of the settings for Monarch have been created, you are now ready to upload with Monarch.

In the main window of Monarch, select either **Files** or a **Folder** from the input selection.

Next, select a destination folder in Onshape. All documents and folders will be created in this destination, inheriting the permissions and access of the parent folder.

When files are selected locally, the list of files will be shown in Monarch. This list does not include any parts referenced in assembly files. Parts referenced by assemblies will be added Onshape, even if they are not in the preview list.

Parts that could be ignored via The ignore files rules will still show up in the preview, but will not show up in Onshape.

Assemblies

Assemblies in SolidWorks are constructed from imported parts. The assembly will be reconstructed in Onshape by importing parts into an assembly studio. Parts are placed in the assembly in their current position in SolidWorks. If the option is selected, the assembly will then be grouped in the “as imported” group. Mates and other assembly features are not supported.

Suppression is read from SolidWorks and replicated in Onshape. Exploded views and configurations and cause issues with this. The Misc Options setting “Unsuppress all components” will leave all assembly items as unsuppressed.

Related

Non-CAD Files that match a CAD file as described in the Non-CAD file settings page will be uploaded to the same document as the related CAD file.

Configurations

Each configuration of a part or assembly is created in a separate studio in the part/assembly document.

The name of the element (tab) will have the configuration added to it.

Versioning

When a part or assembly has been uploaded, a version will be created. This version will be referenced in assemblies that are recreated. The version will have the description “Created by Monarch” to indicate a first version.

If a related file is uploaded to a document on subsequent uploads, an additional version is not created. Versions will only be created on the initial upload.

Along with creating versions, Monarch will cleanup the document to remove the default part studio and assembly.

Folders

If the “create folders” option is selected in the settings, Monarch will replicate local file structure in Onshape. This can be useful to replicate the organizational structure of a PDM vault or local share drive. This option is only available if a source folder is selected, not individual files.

When uploading, the destination folder in Onshape will be treated as the selected local folder. All local sub folders (recursively) will be created in Onshape with the same names. If a folder already exists in Onshape, the existing folder will be used. All files will be uploaded to their corresponding folder in Onshape. If a file has already been uploaded to Onshape in a different location, it will not be moved.

Subsequent Uploads

Monarch saves a database of all files that have been uploaded to Onshape. Monarch references the specific document and element in Onshape, and not the location. If a document is moved to another folder in Onshape, that reference is kept. If an existing part has been added to an upload, it will be skipped by Monarch to prevent duplicates.

Monarch keeps a reference to the first version of a document that is uploaded. If a part is changed in Onshape, and used in a future assembly upload, the originally uploaded part will be referenced in the assembly.

Failed Items / In progress items

If an item fails to upload, or the upload process is cancelled while an item is in progress, Monarch will not be able to recover and continue. These items may exist in Onshape as a partially finished document. Monarch gives users the option to remove these partially finished documents.

In the Monarch Error Log window, the number of incomplete items will be displayed. The button the left will allow the user to remove these incomplete documents. **This will delete the unfinished documents in Onshape that were created by Monarch.**

Incomplete documents are not added to the Monarch Database and left ignored will not have an impact on uploads, but may confuse users. It is recommended to cleanup these documents after any potential issues have been diagnosed.

Updating files using a Csv file

- Monarch can update file properties (Meta data) using a csv file.
- The process is:
- Select the menu option “Update from...”
- Select a *.csv file which uses comma’s as separators.
- The contents of the CSV file must contain the following minimum columns:
 - Document URL
 - Any other property specified by name (case sensitive, space sensitive)

	A	B	C	D	E	F
1	DocumentUrl	Description	Part number	Revision	Vendor	Project
2	https://cad.onshape.com/documents/a8fd219855870258ac3f9c44/w/3afe698c43bb13a45d1a783d/e/9c63a07aac4137f61d023d4/p/JFD	UT desc1	PN-43H0104	A	AK	UT-1
3	https://cad.onshape.com/documents/c759abfcea641ac5b8d2bcd/f/w/703c93cb469ccec3891e7ffa/e/4fe29f10ac5c7e59898a3624	UT desc2	PN-43H0105	A	AK	UT-2
4	https://cad.onshape.com/documents/a8fd219855870258ac3f9c44/w/3afe698c43bb13a45d1a783d/e/9c63a07aac4137f61d023d6	UT desc3	PN-43H0112	A	AK	UT-3
5						
6						

- The name of the property must exactly match that found in the company properties.
- If the name does not exactly match an error will be shown

CADSharp company settings > Properties

Users

Teams

Properties

Categories

Preferences

Items

Applications

Numbering schemes

These properties only apply to documents owned by CADSharp.

Create custom property ?

Property

Appearance

Name






Description

Category

Part number

The following properties are not yet supported:

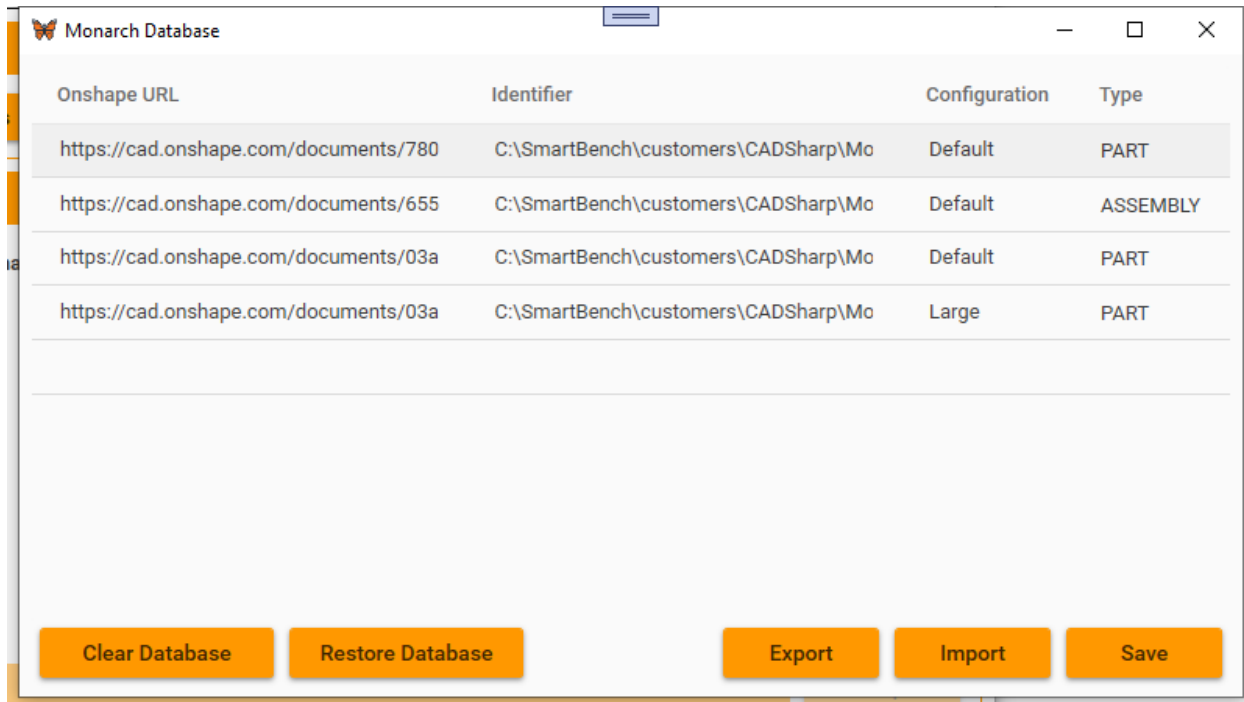
- Material
- Category
- Any properties that are of type ‘object’

Property	Type	Category
 Appearance	➔ Object	Onshape Part
 Name	Text	Onshape Global
 Description	Text	Onshape Global
 Category	➔ Category	Onshape Global
 Part number	Text	Onshape Applicatic

Monarch Database

The relation between SolidWorks files and documents in Onshape is stored in a local database on the User's computer. This database relates the Onshape URL to the uniqueness value as described in the Monarch Settings page for file uniqueness.

It is recommended to backup this database regularly.



Onshape URL	Identifier	Configuration	Type
https://cad.onshape.com/documents/780	C:\SmartBench\customers\CADSharp\Mo	Default	PART
https://cad.onshape.com/documents/655	C:\SmartBench\customers\CADSharp\Mo	Default	ASSEMBLY
https://cad.onshape.com/documents/03a	C:\SmartBench\customers\CADSharp\Mo	Default	PART
https://cad.onshape.com/documents/03a	C:\SmartBench\customers\CADSharp\Mo	Large	PART

Clear Database Restore Database Export Import Save

Exporting Monarch's Database

Backing up or exporting Monarch's database is simple.

In the database window, click the export button, and choose a reasonable location and file name for the database. We highly recommend backing this database up to a cloud-based storage solution. You can even upload it to a document in Onshape.

To export as CSV, specify a file path with the CSV extension.

Example: "C://mydatabase.csv"

It is recommended to export as a tab separated file ".tsv".

The reason is that file names can contain comma's and therefore *.tsv files are a 'safer' export option to use.

To Export as TSV, specify a file path with the TSV extension.

Example: "C://database.tsv".

This prevents issues with commas in filenames.

Working on multiple computers

The database that Monarch works with is local to the computer it is run on. In the case of running Monarch on multiple computers, the database can be exported, and then either imported or merged.

Importing the Monarch database replaces the current database.

Merging the databases will add the current database to the imported database. Any duplicates are removed.

Keep in mind that when using file path to define file uniqueness, the entire file path is considered. It is recommended that a property, such as part number, is used to define uniqueness when using Monarch across multiple computers.

Editing the Database

If files were uploaded without Monarch, those documents and elements can still be added to the Monarch database. Adding new rows in the Monarch database window, filling in the necessary information, and clicking “save” will update Monarch’s database for future uploads.

Importing into the database

Two options are available for importing into the database. The first option is to import a previously exported database.

The second option is to import a CSV file. The CSV file must be in the format of:
Onshape URL, Uniqueness, configuration, type.

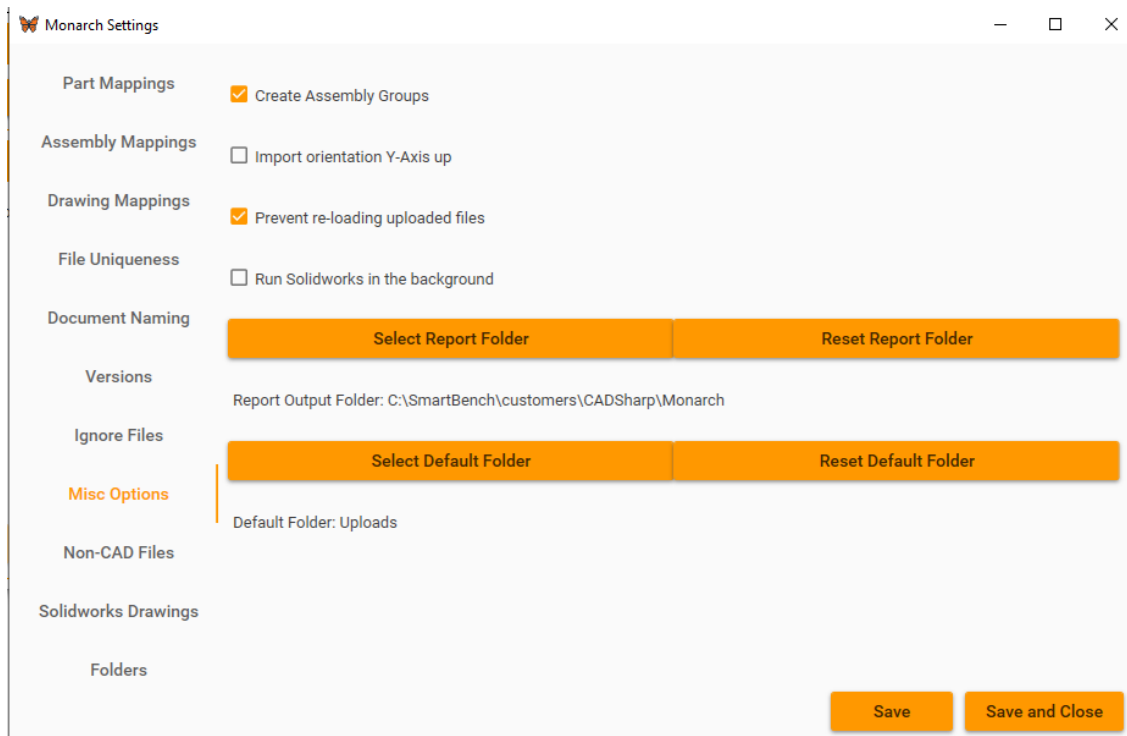
This format matches the format from the csv export option.

The header in the CSV is optional.

The Onshape URL may contain a configuration but should not contain other URL parameters.

Upload Reports

Upload reports will be generated automatically if a report output folder is selected. Upload reports contain information of files loaded, files throwing errors, skipped files, and file upload information.



Monarch Settings

- Part Mappings**
 - ☒ Create Assembly Groups
- Assembly Mappings**
 - ☐ Import orientation Y-Axis up
- Drawing Mappings**
 - ☒ Prevent re-loading uploaded files
- File Uniqueness**
 - ☐ Run Solidworks in the background
- Document Naming**
 - Select Report Folder
 - Reset Report Folder
- Versions**
 - Report Output Folder: C:\SmartBench\customers\CADSharp\Monarch
- Ignore Files**
 - Select Default Folder
 - Reset Default Folder
- Misc Options**
 - Default Folder: Uploads
- Non-CAD Files**
- Solidworks Drawings**
- Folders**

Save Save and Close

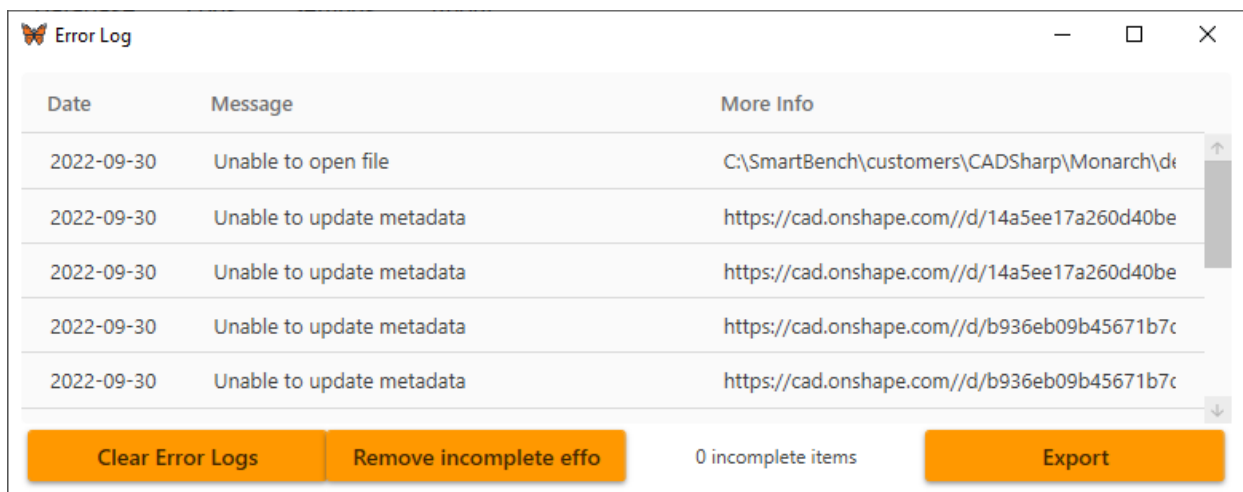
Monarch Error Log

If Monarch encounters an issue with a file, the file is skipped and logged in the error log.

When complete, Monarch will display a message to the user indicating that errors have occurred.

Error can be seen in the “Error log” section of Monarch.

The error log file contains more details than shown. When reporting errors to CADSharp support, please export the error log file and include it, along with the date and time of the error.



Date	Message	More Info
2022-09-30	Unable to open file	C:\SmartBench\customers\CADSharp\Monarch\di
2022-09-30	Unable to update metadata	https://cad.onshape.com//d/14a5ee17a260d40be
2022-09-30	Unable to update metadata	https://cad.onshape.com//d/14a5ee17a260d40be
2022-09-30	Unable to update metadata	https://cad.onshape.com//d/b936eb09b45671b7c
2022-09-30	Unable to update metadata	https://cad.onshape.com//d/b936eb09b45671b7c

Clear Error Logs Remove incomplete effo 0 incomplete items Export

Common Issues

All of my parts are incorrectly positioned in my assembly.

You may need to use the setting “Y-Axis up”.

Monarch is stuck on “Starting Upload”

This could be an issue with SolidWorks. If SolidWorks has any install errors (such as *journal file not found*), this will prevent Monarch from accessing SolidWorks.

Is SolidWorks Connected supported?

Currently SolidWorks Connected is not supported. Only SolidWorks versions that can save to the local computer are supported by Monarch.

Support

Please email all questions and issues to keith@cadsharp.com with a subject line containing “Monarch”.

If an error has occurred, please make a note of the date and time, along with the email address of the Monarch user.