



What's New

in PowerPack for Inventor 2023



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1. Generalities Version 2023

The 2023 version of the **PowerPack for Inventor** comes with new features but also improvements on the existing ones:

- Read the Inventor Version
- Remove iLogic Rules
- Border Frame Swap

The users of Autodesk Inventor versions 2022 and 2023 will benefit from these novelties from the **PowerPack for Inventor 2023** because this last one can be installed on both versions of the platform.

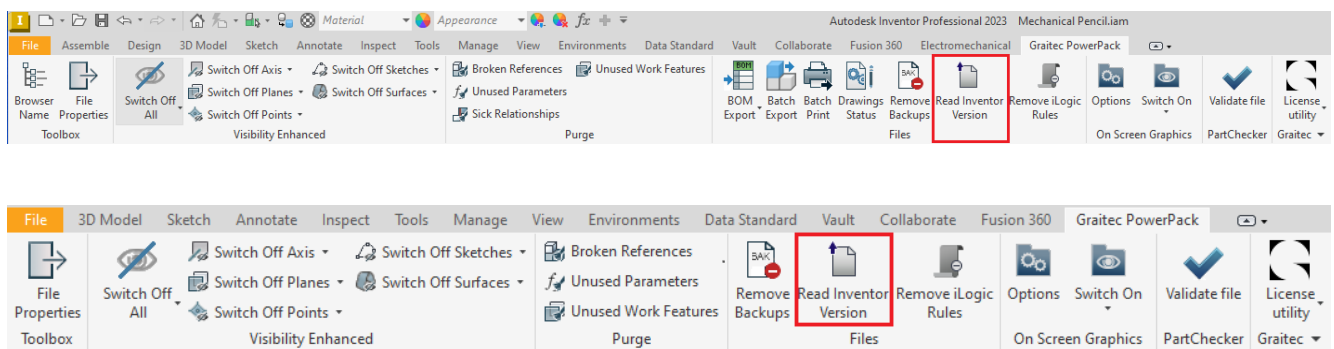
1.1 Read the Inventor Version

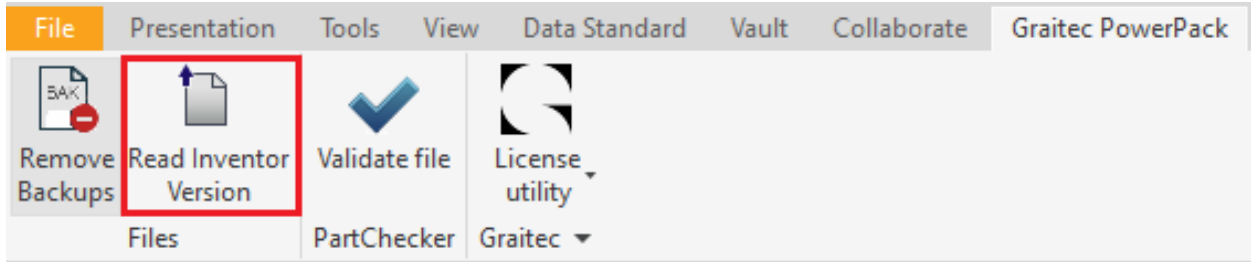
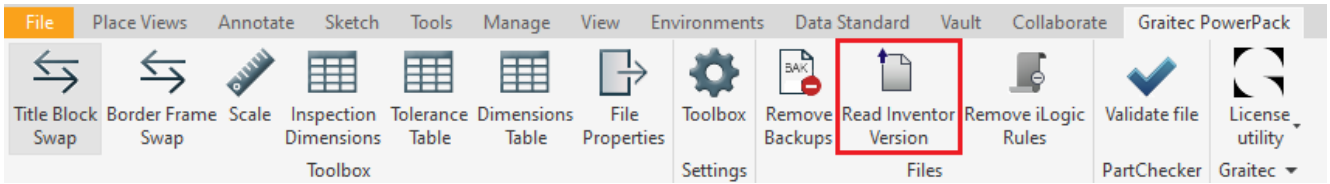
Main features & benefits:

- Reduce manual checking of the parts and assembly
- Search in all the model
- Migrate the files from old version to latest version
- Possibility to use the function on part, assembly, drawing or/and presentation

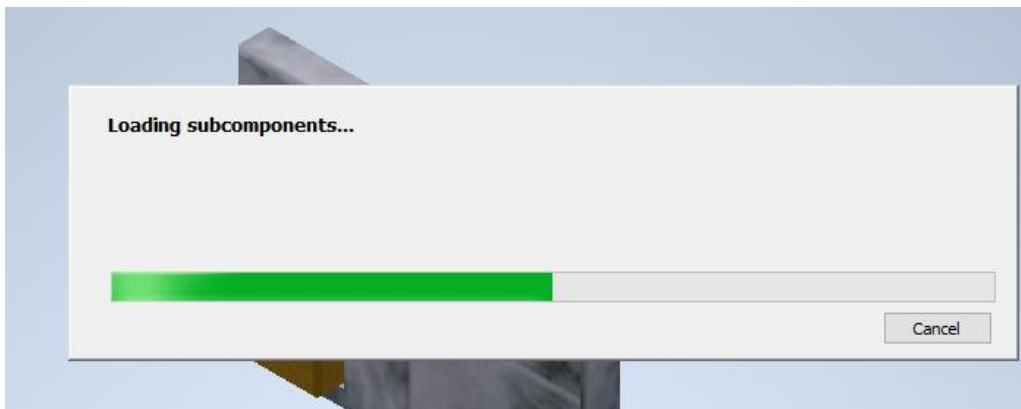
The **Read Inventor Version** is the tool to identify the file's version and update them to the latest because up-to-date Inventor files are important for stable and performant CAD.

The functionality can be accessed from the GRAITEC PowerPack ribbon, under the **Files** category, no matter the environment part, assembly, drawing, or presentation.



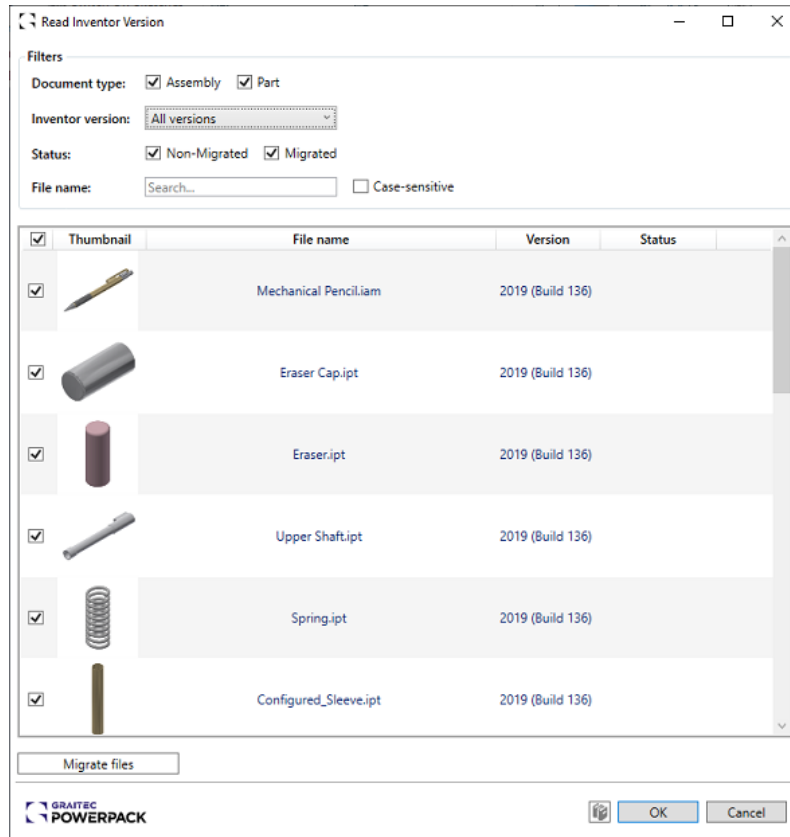


When accessing the command, an automatic search is done in the entire project. The command is searching and loads all the Inventor files that are currently opened.



Loading files

The files are listed in a dialog where multiple operations are possible.



Read Inventor version dialog

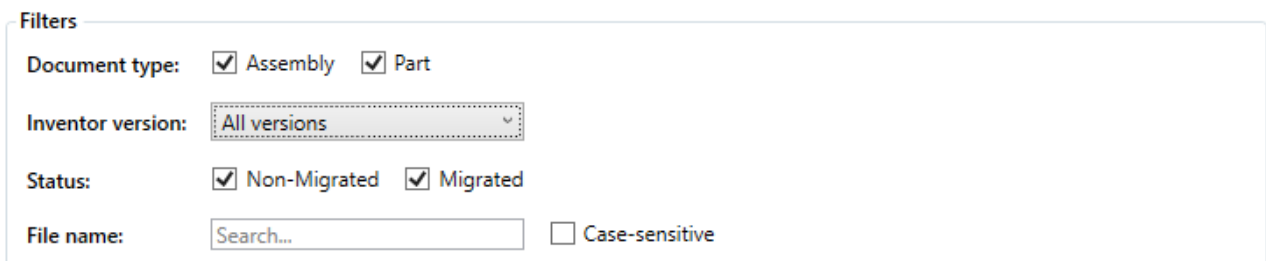
The **Filters** are making the selection of files much easier.

The user can select only assemblies, or only parts, or both.

Also, an option to filter by the **Inventor version** is available as well.

A filter by **Status** is also available, offering the possibility to show non-migrated or migrated files.

And in case of the need to migrate a particular file, it's very easy to filter it by name, thanks to the **File name** option.



Read Inventor version filters

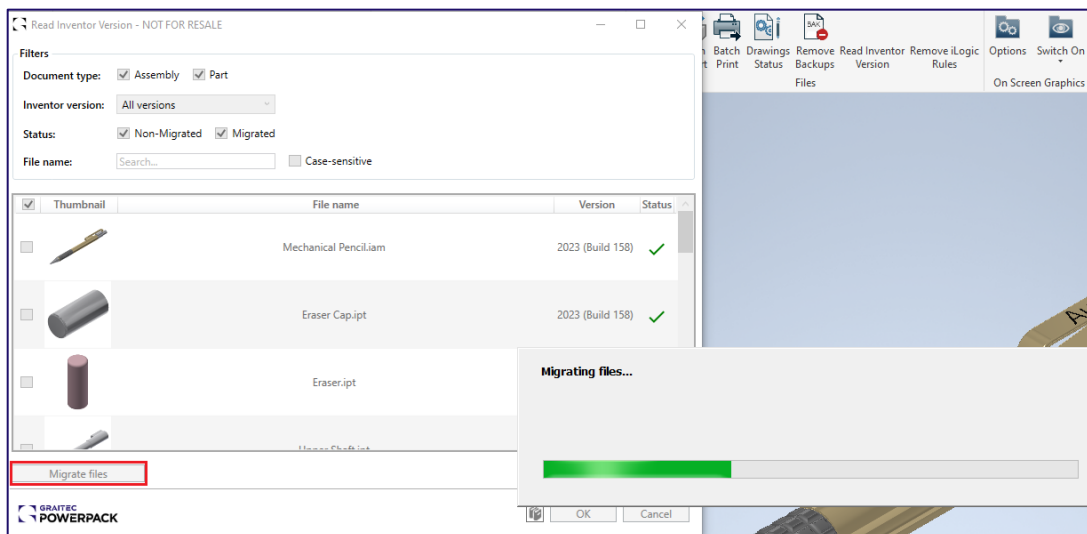
After the filters, a table is present, a table in which the user can:

- Select one or more files/rules – also an option of selection or deselection of all listed elements is available
- See exactly what is in the files thanks to the image present in the table
- Identify the detected files by their names
- See the Version of each file loaded, including the build number
- The status: if a green check appears, the file is migrated to the latest version of Inventor. If no check is provided the file is not yet migrated.

Selection	Image of the object	Detected files	Current File Version	Status Migrated or Not
<input checked="" type="checkbox"/>		File name	Version	Status
<input checked="" type="checkbox"/>		Mechanical Pencil.iam	2019 (Build 136)	
<input checked="" type="checkbox"/>		Eraser Cap.ipt	2019 (Build 136)	
<input checked="" type="checkbox"/>		Eraser.ipt	2019 (Build 136)	
<input checked="" type="checkbox"/>		Upper Shaft.ipt	2019 (Build 136)	
<input checked="" type="checkbox"/>		Spring.ipt	2019 (Build 136)	
<input checked="" type="checkbox"/>		Configured_Sleeve.ipt	2019 (Build 136)	

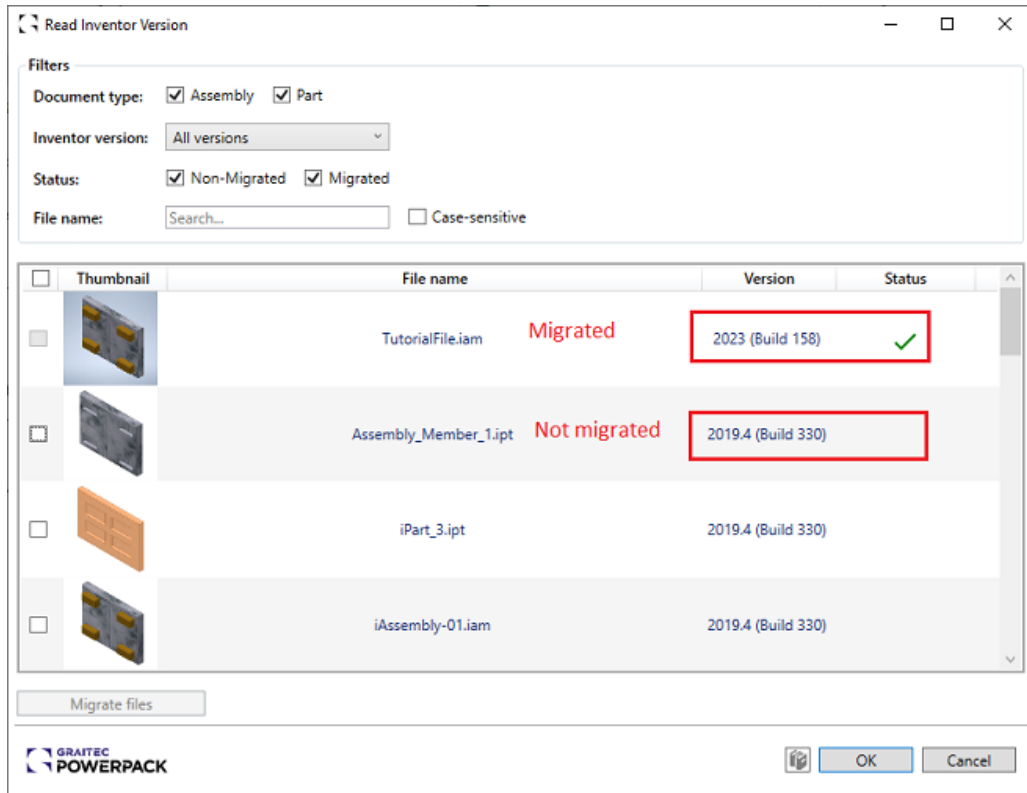
Read Inventor version table

Once the user has identified the files he needs to migrate, by pressing **Migrate Files** button the process is starting and a progress bar appears to follow easy the progression of the migration.



Migration progress

NOTE: *The time of migration is highly linked with the number of files selected for migration.*



Migrated – Non-migrated files

1.2 Remove iLogic rules

Main features & benefits

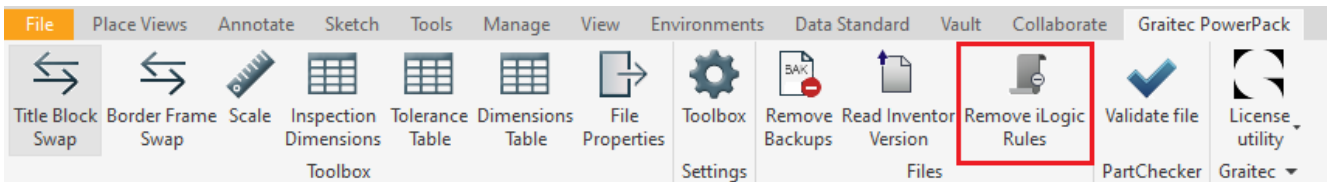
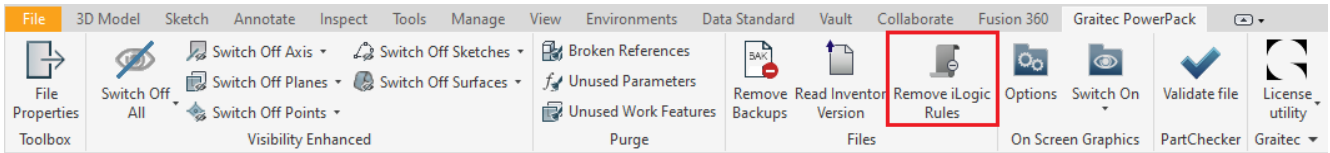
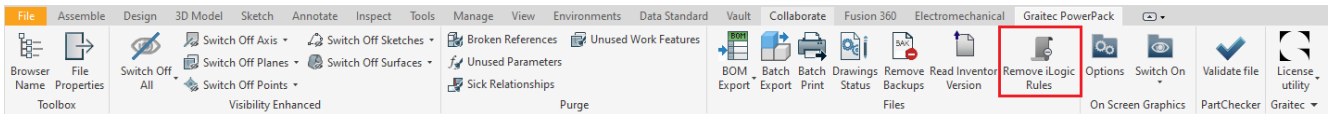
- Reduce manual cleaning of the parts and assembly
- Search in all the model
- Remove the iLogic rules on mass for all the components
- Possibility to use the function on part, assembly, drawing

The **Remove iLogic rules** command has been created to help all designers who have worked with iLogic previously.

Let's imagine a designer has 1000 old files and in individual files, there is a specific iLogic rule for each file.

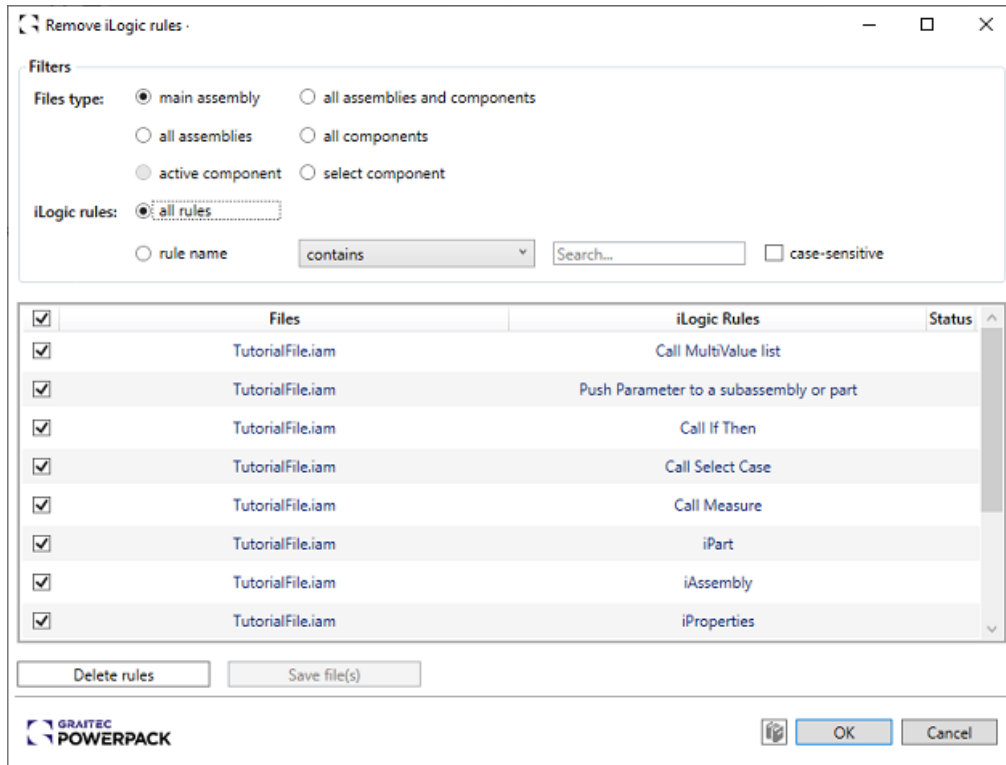
Now, how to centrally manage the rules via an external rule and clean up the iLogic rule within individual files? With the new command from PowerPack for Inventor.

Available in the Graitec PowerPack ribbon, under the files category, the command can be accessed from the assembly ribbon, the parts ribbon, or the drawing ribbon.



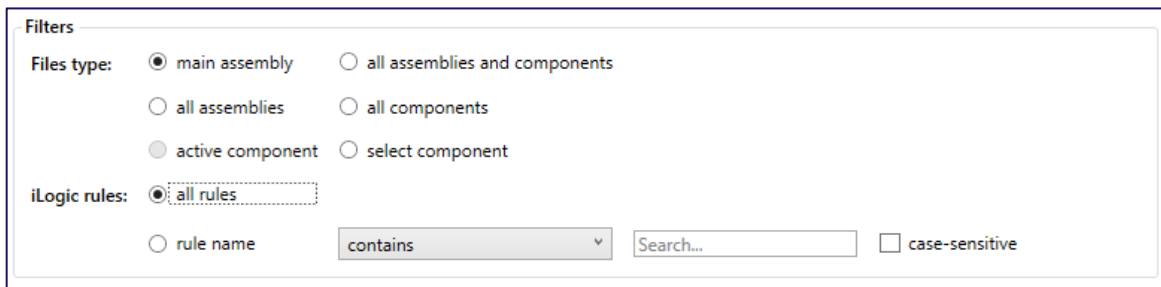
When accessing the command, an automatic search is done in the entire project.

The command is searching for files with **iLogic rules**. If a file does not have **iLogic rule**, then this will not be listed in the dialog that is loaded after the search is finished.



Remove iLogic Rules dialog

The dialog is offering the possibility to apply some filters, to find faster the files which are needed to remove the **iLogic rules**.



Filters area

There are 2 types of filters:

- For elements of the project – assemblies and components
- For iLogic rules

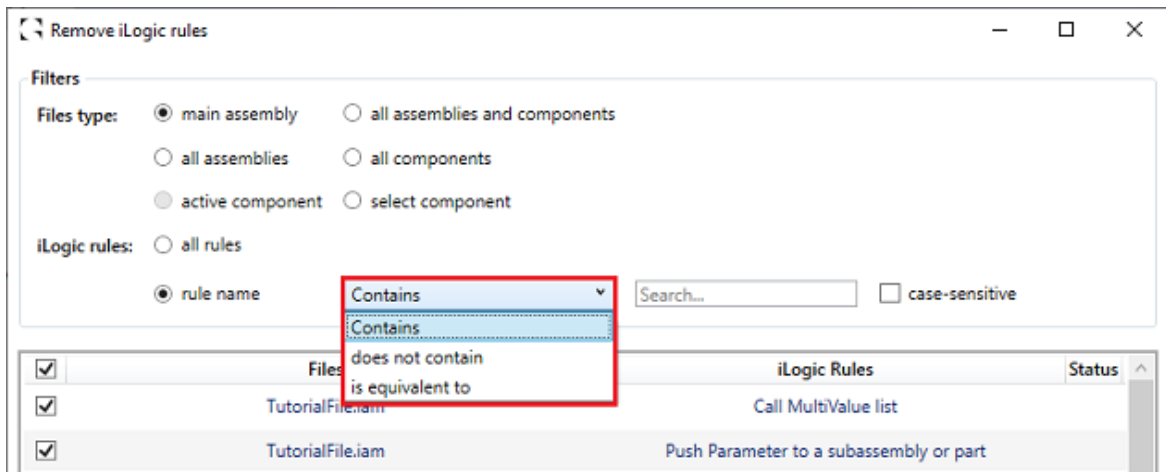
The filters available for assemblies and components are:

- Main assembly – is filtering only the present assembly loaded
- All assemblies – is listing all assemblies loaded in the application
- All assemblies and components – is filtering all the assemblies and their components
- All components – is listing all components loaded in the application

- Active component – is loading just the active component
- Select component - is allowing the selection of components live within the construction area or the browser

The filter for iLogic rules offers the possibility to:

- Select all the rules
- Select a specific one, by name, allowing the definition of a certain name/word present, not present, or identical with the name of the rule.



iLogic rule filter

After the filters, a table is present, a table in which the user can:

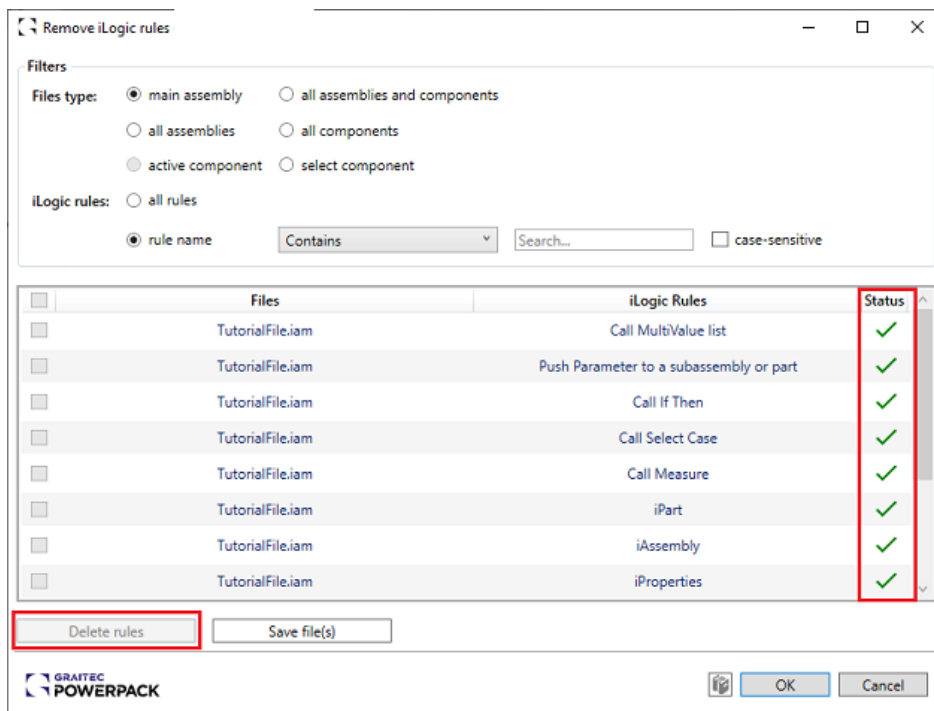
- Select one or more files/rules – also an option of selection or deselection of all elements listed is available
- See the files which are selected for the operation of removing rules
- The iLogic rules available within each file
- The status: if a green check appears, the rule is deleted for the specific file. If no check is provided the rules are not yet deleted.

Selection Files loaded Detected rules based on the filters Status of the rule, deleted or not

Selection	Files loaded	Detected rules based on the filters	Status of the rule, deleted or not
<input checked="" type="checkbox"/>	Files	iLogic Rules	Status
<input checked="" type="checkbox"/>	TutorialFile.iam	Call MultiValue list	
<input checked="" type="checkbox"/>	TutorialFile.iam	Push Parameter to a subassembly or part	
<input checked="" type="checkbox"/>	TutorialFile.iam	Call If Then	
<input checked="" type="checkbox"/>	TutorialFile.iam	Call Select Case	
<input checked="" type="checkbox"/>	TutorialFile.iam	Call Measure	
<input checked="" type="checkbox"/>	TutorialFile.iam	iPart	
<input checked="" type="checkbox"/>	TutorialFile.iam	iAssembly	
<input checked="" type="checkbox"/>	TutorialFile.iam	iProperties	

Table – selection, rules, files, and status

Once the **Delete rules** button is pressed, the rules are deleted, the **Status** is changed and the **Save file(s)** option is activated.



Status for deleted rules

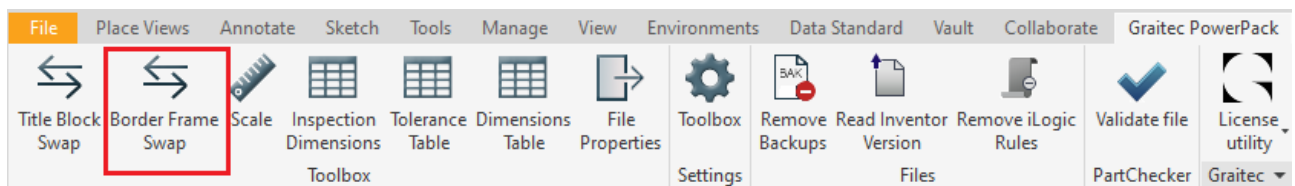
1.3 Border Frame Swap

Main features & Benefits:

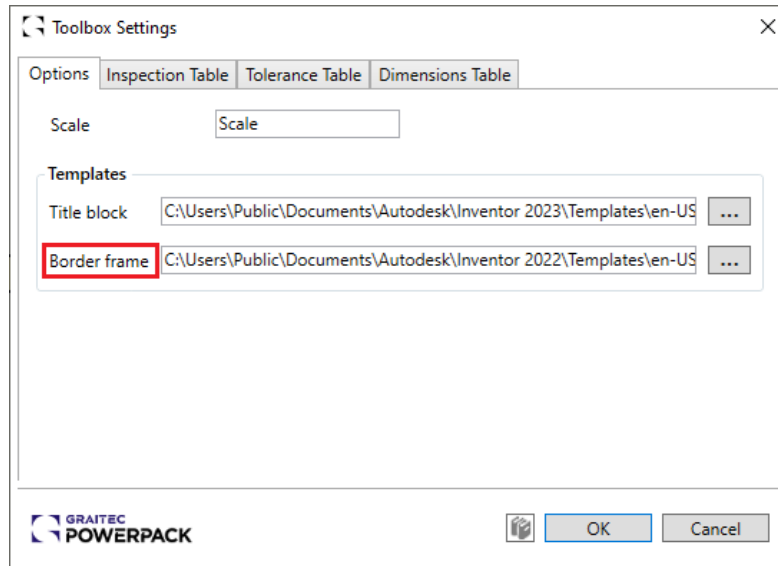
- Easy way to configure the DWGs border
- Independent option to swap the border frame
- Easy definition of the template
- Change fast the DWG

The **Border Frame Swap** command comes to complete an existing workflow for drawings, the possibility to change easily title block in drawings.

Available next to the **Title Block Swap** command in the Graitec PowerPack ribbon dedicated to drawings, the **Border Swap** command helps to change with one click the border of the dwg according to the template set by the user.

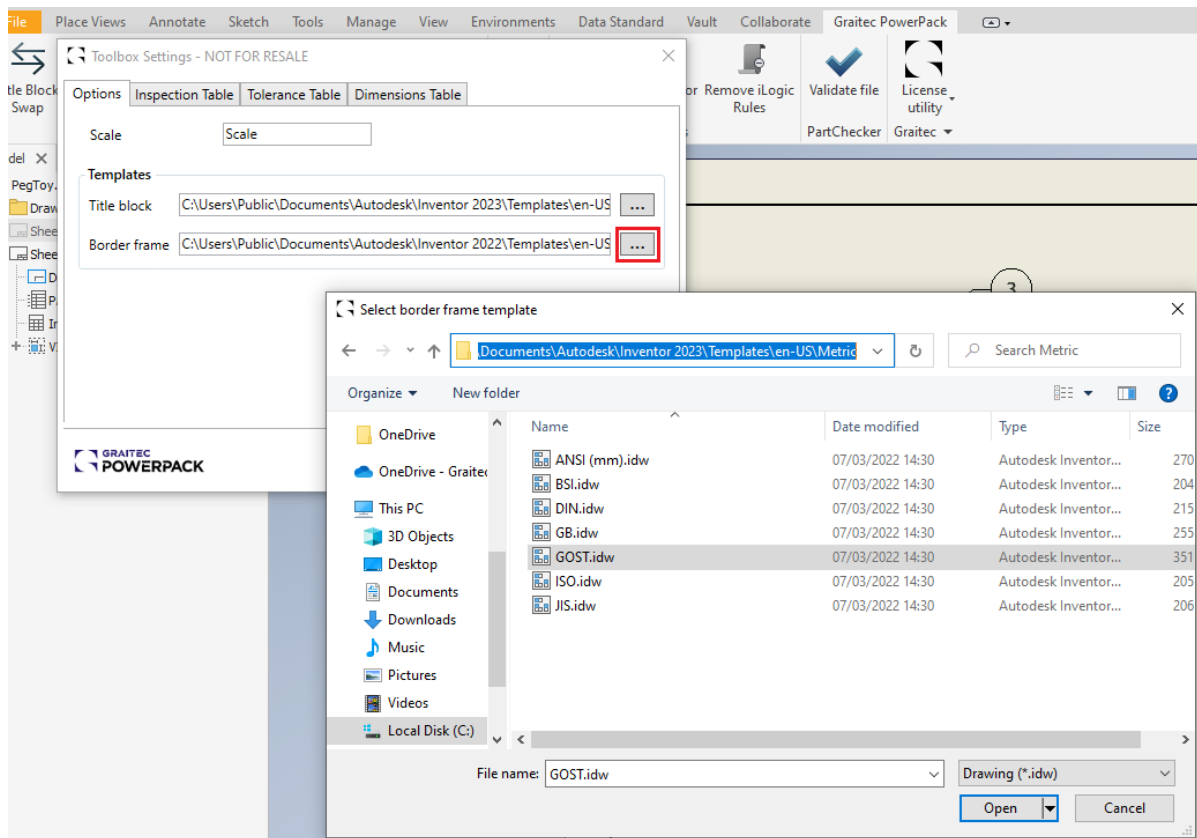


Within the **Toolbox settings** dialog box, an additional option was added to select the template file which has the border required for the swap out.



Toolbox settings

By pressing the 3 points icon for the border frame, the user will be allowed to define the template border used when the new command is run.



Select template border