Tool Full Manager

File based saving of a resource structure in CATIA

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1. Foreword

This description is for Version 1.0

This tool is used to manage the files loaded in the CATIA. Helps you to maintain order, search for names and change them to selected phrases. One of the main advantages of this device is the addition or changing of selected names to the items being selecting or just for all items. You can use this tool if you have installed Catia Version 5-6 Released after 2014 with Service Pack 4, Build 24. There is no required to have installed any special HOT FIX.

This tool works only on the Windows Operating System. It is recommended to use Windows 7 Professional. Your computer must have installed library .NET Framework version 4.0 or upper. Additional library is taken out from CATIA library.

This description is for Version 1.1

New option is available. Now you can add prefix or suffix to all or selected part numbers. In code there is option to check before conflict with system naming.

This description is for Version 1.2

New Design of Tool was implemented. Now you can move split sign between columns, colors were change to system and similar background like in CATIA.

This description is for Version 1.3

New option is available. Now you can add numbers to all or selected part numbers. In code there is option to check before conflict with system naming. Also you can use option to delete all pieces of text with phrase Copy (…) of.

This description is for Version 1.4

New option is available. Console Tab contain information about everything what was done in CATIA and in Tool.

This description is for Version 1.5

New shortcuts were added

This description is for Version 1.6

Improving code transparency. Changing the method of calling some methods, including implementing new classes and using the constructor to update the application refresh icon.

This description is for Version 1.7

New design, size and colors. Update documentation.

2. History

16.08.2017   Starting the tool (CATIA-Full Manager).

20.08.2017   New option is available such as Adding Prefix / Suffix.

28.08.2017   New Design of Tool.

21.09.2017   New option is available such as Adding Numbers and Delete "Copy of".

02.10.2017   New option is available: Console Tab.

05.10.2017   New shortcuts were added.

13.08.2019   Update all layout of application, colors and documentation.

3. Definition

The tool is designed to manage files with their name Part Number, the name of the files that are open in CATIA and instance names.

The basic requirements are as follows:

1. Saving a BEMI structure (Save)
2. Renaming individual components of a BEMI structure
3. Rename an entire BEMI structure
4. Search and replace specific phrases in names
5. Change all instances names in open documents
6. Adding prefix and suffix to names.
7. Change File Name As Part Number.
8. Adding special counting number after names
9. Delete special Phrase Copy of after using break link option.

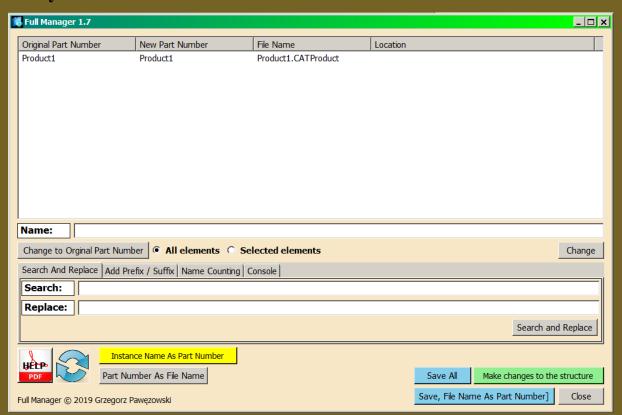Some of the main constraints/requirements are as follows:

- You can run tool only at active document which must be a Product.
- Part Number is individual name in all documents opened in CATIA, so to be sure there is no way to change Part Number of two different documents to the same
- Tool working for all documents opened in CATIA not only for Active Window in CATIA.
- After using button called "Save All according to Part Number" there is no way to undo changes
- Make sure before using button called "Change Instance Name according to Part Number", because it's working only for elements which is connected with documents loaded in CATIA.

WARNING!

Be careful when delete or add new products, parts during tool "Full Manager" is working. If you want use Tool after this operations, just use Refresh Option.

If you add some new components, then use undo option part will disappear, but doesn't disappear from you memory! It's still in CATIA cache. Remember this, because for now Full Manager is checking all cache.

4. Layout

5. Main functions

5.1 List of components in CATIA.

      This list contains all opened documents in CATIA that will be changed. From the left to right, first column contain Original Part Number, elements cannot be changes during tool's lifetime. Next column "New Part Number" includes all changes which you want to implement to your Assembly in CATIA. Third column contain File Name connected with Element from left. The last column presents disc location for files from left column. Remember changes showed in second column will be implemented to Catia only after entering button "Makes changes to the structure".

5.2 Selection Mode Option.

Main option, this Mode Option presents a working process of tool.

1. "All elements" option checked means you are working on all elements which you see in List above.
2. "Selected elements" option checked means you are working only on that's elements which you have checked in List above.
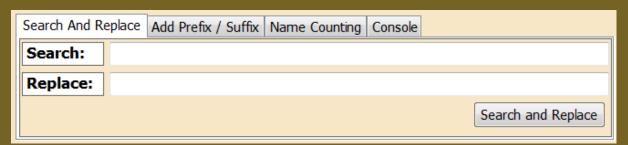
5.3 Change Part Number.

      Space after "Name:" it's place to enter your new part number for selected element from List above. Remember this option is available only in All elements mode.

      To specify which element part number you want to change just highlight one from List above. After this you can change name and confirm your decision by clicking on button "Change" or use shortcut "Enter".

5.4 Search and Replace.

Space after "Search:" it's place to enter the phrase which you want to find.

Space after "Replace:" it's place to enter the phrase which you want to replace, if you put black here it's will still be working unless there will be conflict in naming rules.

You can confirm your decision by clicking on button "Search and Replace" or use shortcut "Enter" after entering "Search:" or "Replace:" value.

You can active tab using the shortcut "F1".

5.5 Adding Prefix or Suffix.

Space after "Prefix:" it's place to enter the phrase which you want add as a prefix.
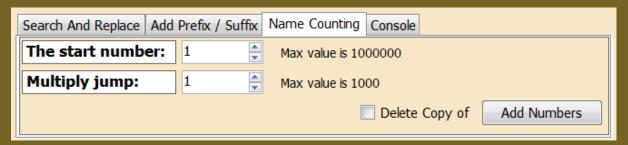
Space after "Suffix:" it's place to enter the phrase which you want add as a suffix.

Always program checking for conflict in naming rules, so you don't have to worry about this.

You can confirm your decision by clicking button "Add Prefix/Suffix" or use shortcut "Enter" after entering "Prefix:" or "Suffix:" value.

You can active tab using the shortcut "F2".

5.6 Adding Name counting + option to delete Copy of.

Space after "The start number:" it's place to enter the number from you want to start your counting elements. Remember it's not available to start your number with "0", you can add this simply by Add Prefix/Suffix Option. Like you can see the maximum value is 1000000, if you want more contact with author.

Space after "Multiply jump:" it's place to enter the number, which will be counted next items. Like you can see the maximum value is 1000, if you want more contact with author.

Example of counting:

1 and 2) Before change:

Part1_

Part2_

1) [The start number:] **100** [Multiply jump]: **1**

Part1_100

Part2_101

   2) [The start number:] **101** [Multiply jump]: **2**

Part1_0101

Part2_0103

   If "Delete Copy of" Option is checked, all elements which have this part of name inside part number will be deleted and replaced by specified values enter in "Name Counting" tab page.

   3) Before change: [I add "_" before use option to delete copy of]

Part1

Copy (1) of Part1_

Copy (20) of Part1_

Copy (1) of Copy (2) of Part4_

Copy (1) of Copy (2) of Copy (1) of Copy (12) of Copy (1) of Part1_

   3) [The start number:] **101** [Multiply jump]: **2**
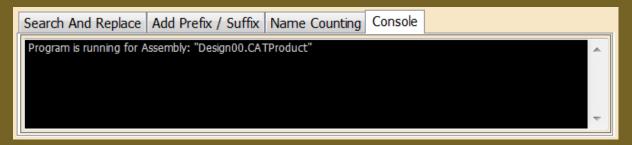
Part1_101

Part1_103

Part1_105

Part4_107

Part1_109

   You can confirm your decision by clicking button "Add Numbers" or use shortcut "Enter" after entering "Multiply jump:" value.

   You can active tab using the shortcut "F3".

5.7 Console.

   The Console tab represents all changes made to CATIA, all warnings if you cannot do some changes and information about current status of tool. You can simply copy and paste every line from this field. Additionally you can export all history by shortcut "E" which create a simply notepad directly in main assembly location.

   If there will be come changes in console, but you don't have active Console tab you will see special icon with bell. This option helps you when you have warning during implements some changes.

You can active tab using the shortcut "F4".

5.8 Change to Original Part Number.

This button allows you to return your part numbers before every changes, remember to implement this to CATIA if you want have the same part numbers inside. This not

5.9 Change Instance Name As Part Number.

This button allows you to change Instance Name to the same As Part Number but with numbers begin of 1. Remember to us this on all Assembly because you can lose some connections from other assemblies.

5.10 Change Part Number As File name.

This button allows you to change Part Number to the same As File Name. Remember this option is available only in All elements mode.

5.11 Make changes to the structure.

This button enters all changes to the items name that were entered in the List above from the "New Part Number" column.

5.12 Save, File Name As Part Number.

This button will save all selected elements and will enter the file name to the same as their Part Number. Remember if you save Assemblies, all elements inside will be saved automatically, there is no independent save option, this works only for Parts.

5.13 Save all.

This button will save all not saved jet elements in CATIA.

5.14 Refresh

This button allows you restart Tool Full Manager, use this option if you change something in structure in CATIA and you want to see update in List. You can run it using the shortcut "Ctrl + R".
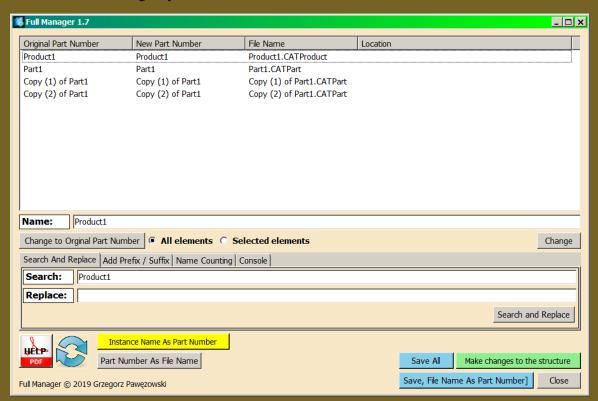
5.15 Help

This button allows you to start this pdf file with all information about tool. You can active tab using the shortcut "F10".
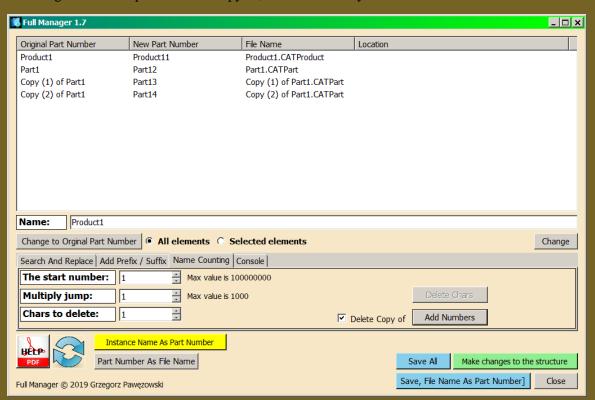
6. Sample procedures

For example if I make some copy of parts and after use the same part as a reference for new component. To do this you need paste special with option "break special".

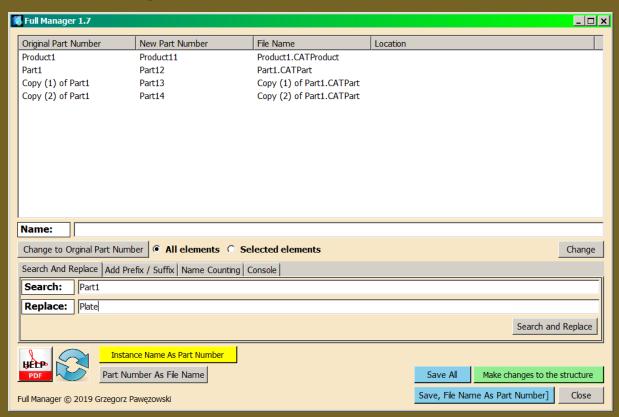This one will give you result showed below:

First thing which we would like to change is names for new format. The best option is go to tab Name Counting and select option Delete Copy of, after execution you will see result:
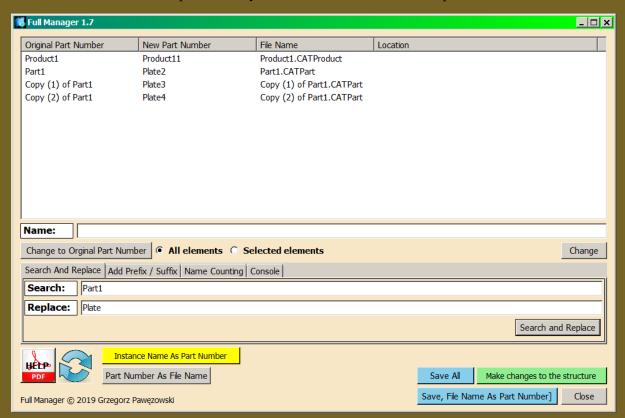
This operation delete all Names of Copy Of, even if we have copy of copy of etc..
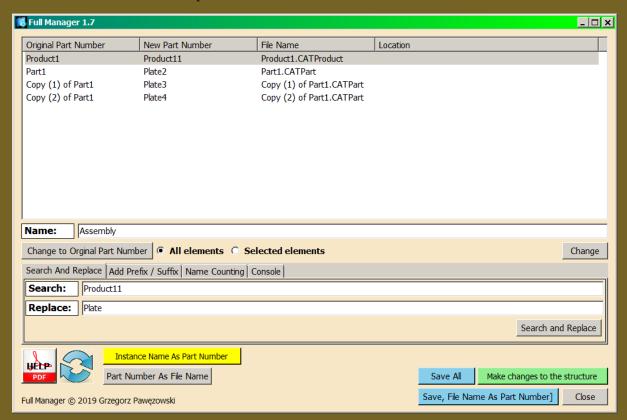
Then we can additional change some part of name for example "Part1" to "Plate". To do this go to first tab "Search and Replace" and fill like below:
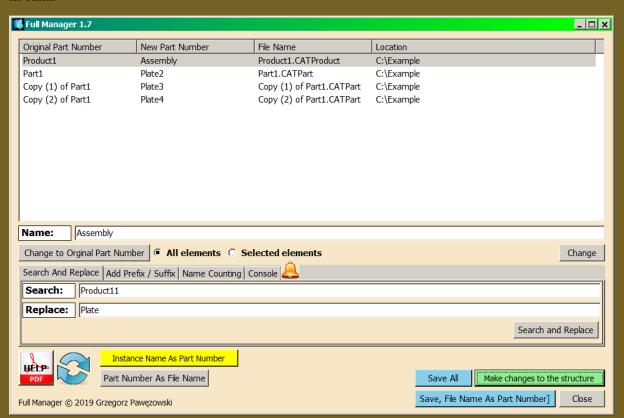

Click button "Search and Replace" then you will see result in List of components.
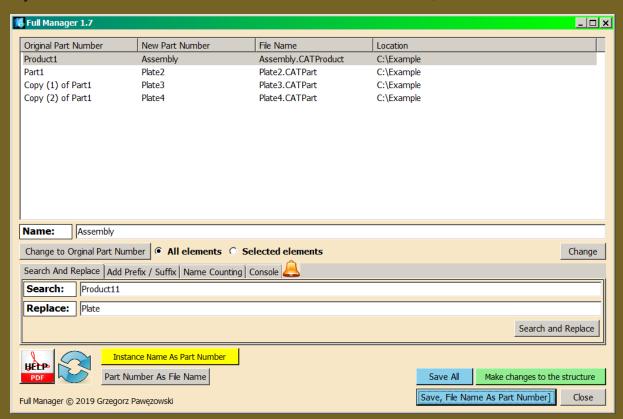
Additional you can change name of "Product1" to "Assembly". To do it you can use Text Box "Name" and select current component like below:

When everything is ready then use button "Make changes to the structure" to update all Part numbers in catia.

If you want have the same File name as a Part number use button "Save, File Name As Part Number"

Last thing to get very clear structure in Catia is change Instance name As Part Number – Yellow button.

Thank you for reading, for any questions contact: gpawezowski@gmail.com