

## How CATIA V5 Works?

Any file you create in CATIA V5 will have the UUID assigned in backend of the software.

UUID stands for a Universal Unique Identifier.

These are 128 bit numbers assigned to any object within a DCE cell (Distributed Computing Environment) which is guaranteed to be unique. The mechanism used to guarantee that UUIDs are Unique is through combinations of hardware addresses, time stamps and random seeds.

There is a reference in the UUID to the hardware address of the first network card on the host which generated the UUID - this reference is intended to ensure the UUID will be unique in space as the MAC address of every network card is assigned by a single global authority and is guaranteed to be unique.

From this it may be observed that the Network address of the ethernet adapter matches the last portion of the uuid. Dassault has made their applications permanently identified with a UUID by creating their Interface Definition Language file in back end of the software.

CATIA identifies the files on the basis of UUID, even if the names are equal This UUID can't be seen or edited in any CATIA function and each CATPart or CATProduct should have such number. Here is a short list of options:

### **1. Create new UUIDs**

- File + new
- File + new from
- File + save As - option save as new document
- INSERT new Product
- INSERT new part

### **2. Keep same UUIDs**

- File + open
- File + save management
- File + save
- File + SaveAs
- File + Send to directory
- File + CLOSE
- File + save ALL.

## How to know a particular CAT file is created in which release of CATIA V5?

Sometimes you could be confused to know in which release of CATIA V5 the files are created for example if you have CATIA V5 R16, R17 , R18. You want to know a particular part file on your desktop is created in which release.

Here the simple Trick to find it.

- 1 . Right click ----> open with Notepad.
2. Go to Edit ---> Find. Now type V5 R16 and enter.
3. If it is mentioned anywhere in the file V5R16 it will show that and you can identify it is made with V5 R16. Similarly you can type V5R17 or V5R18. In edit -- > find option. It can show you on which release you have made the file. And also it shows with what service pack the file was made. It works for CAT part, CAT Product and CAT Drawing.

**Note** : Once you have opened and saved the data from lower release to higher one in V5. you can not open again in lower version. But We have option to use CATIA V5 utilities for Downward compatibility to get the dump solid.

Hope its useful.

Kindly write me your valuable suggestions.

Thank you

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