TECHNIA
ADDNODE GROUP

10 REASONS FOR TRANSITIONING TO CATIA 3DEXPERIENCE

1. Database document management

CATIA 3DEXPERIENCE connects to the 3DEXPERIENCE platform, and as such enjoys all of the benefits that the industry has come to expect from PDM. Version management, modification rights, release control etc. are all present and configurable to the company's needs. This means that in addition to all of the other features described below, CATIA users migrating from file-based CATIA V5 to 3DEXPERIENCE will be able to work much more efficiently. All of the grief associated with managing collaborative working, such as manually copying files, trying to manage revisions and trying not to accidentally overwrite other's work you have loaded in session, is all dispensed with.
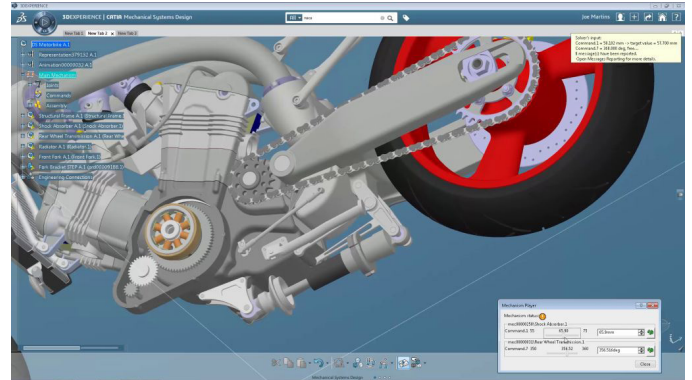
2. Infrastructure changes

Simply put, CATIA 3DEXPERIENCE works quicker by taking advantage of a computer's resources far more effectively than V5, reducing the waiting times on rendering, loading large assemblies and performing complicated mathematics. It may sound like a trivial change to some, but this affects virtually every operation CATIA makes, which can amount to a substantial time saving for every user.

The 3DEXPERIENCE platform is designed to be hosted on the web from the ground up. This means that whether you are connecting CATIA to the 3DEXPERIENCE platform through the web or on the company local network, you will be transmitting and receiving data optimised for the current network state. CATIA will also only request the data it needs when opening assemblies, only loading the full data for the assembly's component parts when requested by the user. This is similar to CATIA V5's cache mode, except that the lightweight files needed for this process are created by a server process beforehand, speeding up the loading process even further.

The web based nature of ENOVIA also makes it easy for users to connect their session to the database, encrypted from anywhere without the need for complex setup on each client machine. This makes collaboration between different sites or businesses convenient.

Note: CATIA 3DEXPERIENCE remains a 'client application', like V5, installed on a user's PC. CATIA 3DEXPERIENCE communicates with the 3DEXPERIENCE platform via an internet-compatible connection. Non-CATIA users connect through a web browser.

3. Searching and exploring the structure in 3D

Even with a PDM system managing data, it is often still onerous on the user to fill in a search form for every part or assembly they wish to search for, particularly when using an external program as mentioned before. CATIA 3DEXPERIENCE however, has an ever-present quick search bar which can be configured to search on any property of the CAD data required (e.g. part number and description). Full and detailed searching is still available should the user wish to construct more elaborate searches.

On top of all of this, the user can also set up lists of favourites and folders to store and organise their own work. Due to the way the database handles the data, designers are free to create any folders or favourites they require without

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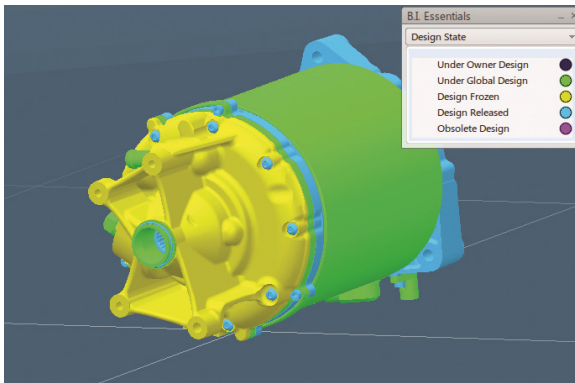
affecting how other users access the data. This is quite unlike using CATIA V5 file based, where moving files into different folders can lead to much frustration, as links between files break as a result of moving and copying them.

All 3DEXPERIENCE rich client applications have the ability to explore the visual structure of assemblies in a quick lightweight mode, without needing to open the full design data. This allows CAD and non-CAD users alike to view the structure of assemblies from the top levels and find the parts they want without needing a powerful computer able to open the top level assemblies in full design mode.

While other PDM systems are able to do something similar, they often rely on CAD users constantly opening and updating the top level assemblies in order to keep the viewer files up to date. The 3DEXPERIENCE platform that controls the data is able to recognise changes to parts and will then automatically update all of the affected representations, keeping all stakeholders up to date in how the structure really looks.

The data stored can be further enhanced by attaching other file types directly to the CAD data (reference tables, exported PDFs, etc.) This documentation is then fully database controlled and is released alongside the CAD data it is attached to.

Finally, it is possible to use the database to visually colour the 3D objects based on various customisable parameters, most notably identifying if objects are released, obsolete or still in work.

4. Integrated database control

CATIA V5 has the ability to integrate with various other PDM systems; however this is typically done with a separate program installed onto CATIA which is used to search for objects and load the required files locally for CATIA to use. Whilst this is functional and certainly an improvement over file-based CATIA, they tend to share a number of limitations with each other, namely:

- The user interface has a line between where the PDM ends and CATIA V5 begins, making for an unintuitive interface which involves swapping between two programs to find, open and save data.

- Complex mappings between CATIA V5 properties and database attributes need to be set up in advance, with the need to synchronise these properties with the database. Given that the user has free reign to modify any CATIA property, such mappings are open to abuse by unscrupulous users.

- Drawings are considered parents of parts and assemblies, meaning that locating the drawing for a part you already have in session requires you to go searching in the database again.

- Multiple revisions of the same part or assembly all must have the same filename, meaning that it is difficult to compare an object with earlier versions, and all but impossible to have one high level assembly with different versions of the same object, since CATIA V5 cannot handle the same filename loaded into session more than once, like most other file-based programs.

- Since the CATIA V5 files need to be stored locally, a user could potentially download the files they want and work outside of database control, send uncontrolled copies of drawings or worse, steal the company's data.

None of these problems are present within the 3DEXPERIENCE platform:

- The user interface in CATIA 3DEXPERIENCE is the database interface and the properties of CATIA 3DEXPERIENCE data are the database properties, so no mappings are required

- Multiple revisions of the same object can exist in session without issue

- Drawings are inserted against the parts and products they represent, meaning that it is easy to find and open drawings for the parts and assemblies you have selected or opened

- Finally, while data is still transferred locally to the CAD machines, it is transferred in a way indecipherable to anything other than that particular CATIA 3DEXPERIENCE session connected to that particular database and permissions to export files in other formats can be controlled per user.
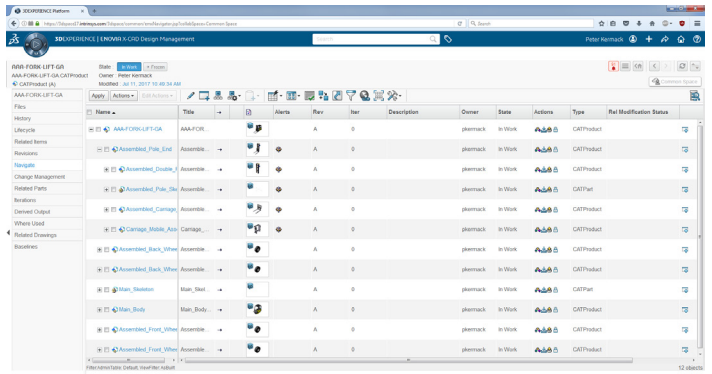
5. Better link management

In file-based operation, a problem invariably faced by any design department is the visibility of links between one CATIA file and the next. These are only visible 'one way', i.e. from the files that are using linked copies of the data to the files where that data was originally created. This makes it impossible to interrogate a file and determine what other Parts, Assemblies and Drawings are currently reliant upon it. One person moving the file or modifying some important geometry can unwittingly ruin any other file that relies on the now-modified part.

Existing database PDM systems do alleviate this problem to some extent by removing the need for the user to move files around, while also enabling the user to identify if links exist between two objects. 3DEXPERIENCE goes one step further by allowing the user to identify the exact features being used and where they are used, from any given object. Geometry links, publication links, contextual links, material links and drawing links can all be identified without the need to open the object up fully into session, meaning non-CAD users can see and gain an appreciation for what parts are reliant on what.

6. New and improved workbenches

CATIA 3DEXPERIENCE features various new Workbenches which exploit the new database functionalities and new graphical capabilities of the CATIA engine. New Workbenches include Natural Sketch, which allows non-CAD users to create concept designs by intuitively drawing on 2D planes in 3D space, creating wireframe geometry which can be directly used by CATIA designers to then create full design specifications.

Existing Workbenches have been modified to directly query the ENOVIA database for its information rather than the local data, meaning that the data you work with is always up to date.

7. 3D BOM management

Typically a PLM system will allow you to create a separate BOM tree from the CATIA structure with the BOM manager overseeing the reconfiguration of this BOM to the real purchased BOM. However they are often using only attributes about the parts, like part number and description, to work with. The 3DEXPERIENCE platform has numerous advantages:

- The CATIA 3DEXPERIENCE designer can declare within CATIA itself which components they have used which are not to be included in the BOM or are phantom assemblies, reducing the amount of time spent querying which parts are necessary in the BOM.

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- The BOM objects themselves are linked to the 3D representations of the CAD models, allowing the BOM manager, or indeed any database viewer with access to the BOM, to view what the parts of the BOM look like in 3D using their web browser. They can even add their own mark-ups to these 3D representations. This reduces the time the user wastes capturing screenshots and creating presentations to share with colleagues.

- The BOM can be compared to other BOMs (such as other versions or variants) in order to compare the two. As well as comparing the tree structure in a table view, the comparison tool also provides a side-by-side 3D viewer of both trees, again in the web page, allowing the user to visually inspect the differences.

8. Proper assembly representation management

In many industries, it is often the case that assemblies are not merely the sum of their parts. Various operations are only performed after the parts have been assembled (drilling new holes, grinding down faces etc). In most CAD systems, CATIA V5 included, it is awkward to represent both how the parts are made individually whilst also showing the finished assembly. This usually involves creating a "Phantom" part to represent the finished assembly.

This is improved in 3DEXPERIENCE where geometry can be inserted directly against the assembly, with a special function called "Derived Representation" which copies all the required geometry of the assembly into one representation. This allows the post-assembly operations to be specified without needing to take out dummy part numbers. Similarly, the ability to insert geometry directly against assemblies removes the need to create "Skeleton" parts to contain construction geometry for assemblies (patterns, package boundaries, centrelines etc).

9. Graphical improvements

As well as various technical improvements, CATIA has also been given a much needed update to its appearance, with the graphical assets re-created to make better use of the high resolution screens that are available today, whilst also reducing visual clutter.

To give you an idea of the update, compare these two trees from V5 and 3DEXPERIENCE. This image also further emphasises how despite the visual changes, a designer would still be able to recognise all of the features of the tree.
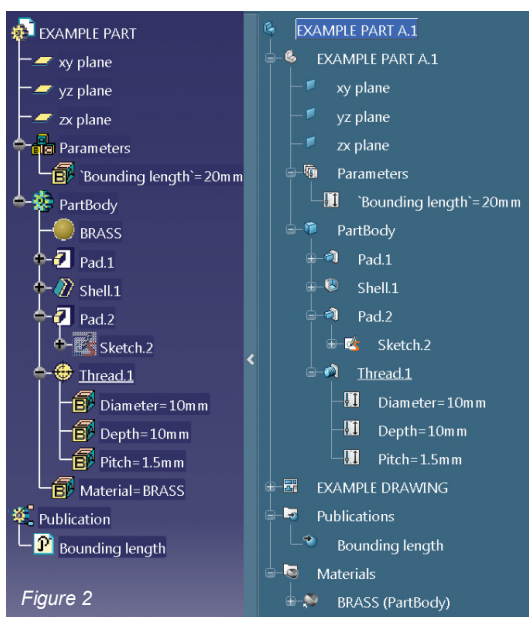

Figure 2

10. CATIA is still CATIA

While fundamental enhancements to the data structure behind CATIA have been made , CATIA's modelling tools remain familiar, with the primary difference being enhancements to the existing tools and additional Workbenches. Users will be able to sit down in front of CATIA 3DEXPERIENCE and be at home without the need for a lengthy transition course.

The underlying kernel for CATIA is based on the V5 kernel; this means that data from V5 can be imported, with full history and links of all kinds, into 3DEXPERIENCE with the database. It has the additional ability to recognise when files have already been imported and re-link these files to any new data, preventing unnecessary duplicates. Imported data can even be later updated with newer versions of the V5 data. Therefore after the import, the users can simply pick up their work where they left off.

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