



Multi-CAD for CATIA V5



INSTALLATION GUIDE

Document Revision: 2.0

Issued: 25/07/2022

Contents

| | |
|--|---|
| Installation Instructions for CATIA V5 Multi-CAD Products..... | 2 |
| Prerequisites..... | 2 |
| Installation Sequence | 2 |
| Post Installation Set-Up..... | 2 |
| Troubleshooting | 6 |
| CATIA Start Up Issues | 6 |
| Insert Existing Component Issues..... | 7 |
| Maintenance Support..... | 7 |



Installation Instructions for CATIA V5 Multi-CAD Products

It is paramount that you install the CATIA V5 Multi-CAD products as the local Administrator user. Failing to do so will result in the installation process not being performed completely. To start the process, download the installation media from the Theorem software download site and double clicking the .msi file. The installation process will then commence.

Prerequisites

Theorem's Multi-CAD products for NX, Creo, JT 3D PDF support have dependencies on the following software:

- The GA level or higher of the supported CATIA V5 version
- Theorem Solutions TXX MultiCAD Platform (for the specific CATIA V5 version used)
- Third Party Application (e.g. NX or Creo installations for supported versions)

For a list of currently supported versions of CATIA V5, NX and Creo please see the Theorem Solutions Product Release Notes document for the appropriate translator.

Installation Sequence

The Dassault Systèmes recommended installation sequence for a successful Multi-CAD installation is as follows:

1. Install required Third Party Applications (e.g. NX or Creo installations)
2. Install the CATIA V5 application at the required Revision at GA level.
3. Install Theorem Solutions TXX Multi-CAD Platform msi.
4. Install any required CATIA V5 Service Packs and Hotfixes.
5. Install Theorem Solutions Multi-CAD Product msi (e.g. this product)

Post Installation Set-Up

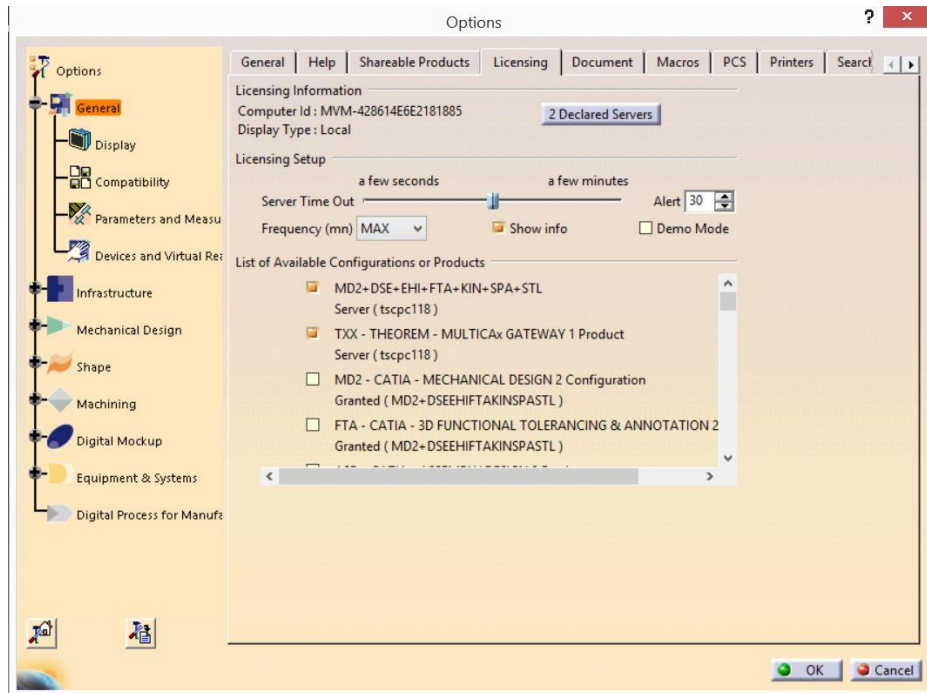
Ensure that the THEOREM_LICENSE_FILE variable is set to point to your license server prior to starting CATIA V5 or the Theorem User Interface. When using the CATIA V5 interface the variable must be set in the active CATEnv file. Alternatively, when using the Theorem User Interface or batch processing operations the variable needs to be set in the ts_env.bat file located in the translator installation directory.

Having carried out the above installation process start CATIA V5 using the desktop icon created upon installation or alternatively copy the CATEnv changes into your existing environment file and start CATIA V5 using your standard start-up process.

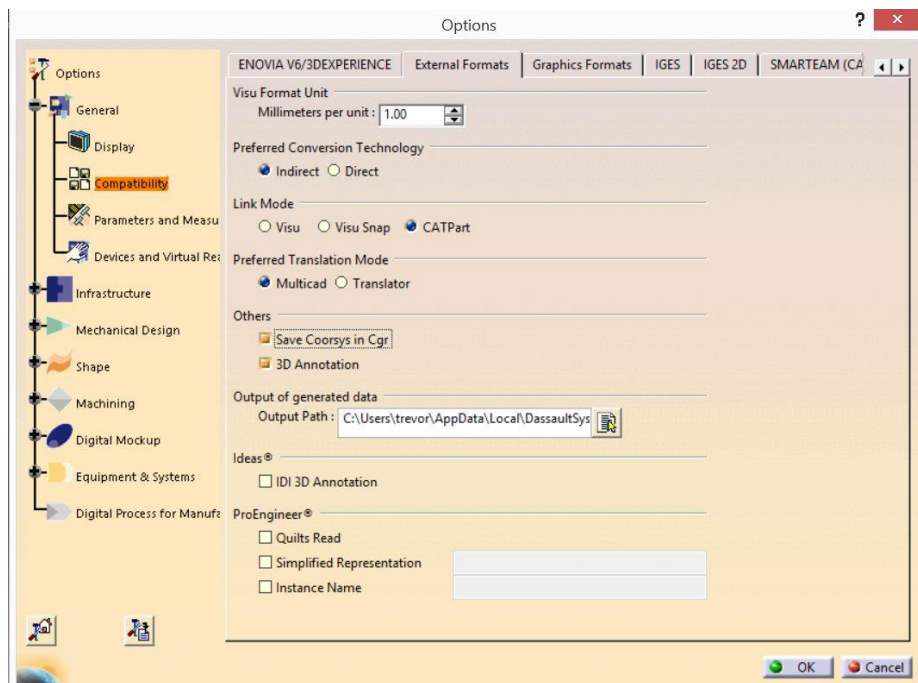
For detailed instructions as to the editing of the CATEnv file please refer to the user guide for the specific Multi-CAD product that you are installing.

In order to use Theorem's Multi-CAD translators, there are 3 set-up requirements:

1. Select a TXX – THEOREM – MULTICAx GATEWAY license via the Licensing tab in the CATIA V5 Tools>Options>General user settings panel.



2. Set the **Preferred Conversion Technology** option to **Indirect** on the External Formats tab of the Tools>Options>Compatibility user settings panel



The user can also specify the output data type via the **Link Mode** option in the External Formats tab. This option controls the format that the inserted geometry will be generated within the V5 output. The choices are;



- **Visu** – Standard CGR representation of input geometry
- **Visu Snap** – Standard CGR + Canonical data representation of the input data. The additional Canonical data will assist with the process of positioning the CGR data by allowing the user to identify planar faces, cylindrical surfaces and centrelines from the CGR geometry
- **CATPart** – Standard BREP CATPart representation of input geometry

The **Preferred Translation Mode** can be set to be either **Multicad** or **Translator** mode.

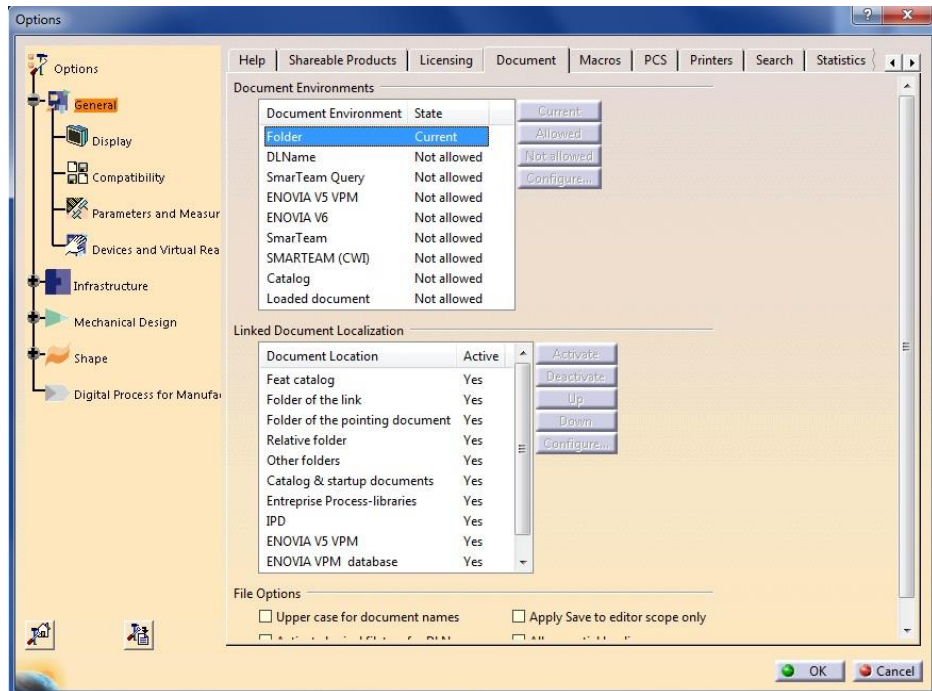
- **Multicad** - When selecting Multicad mode any CATPart's that are created will not be able to be edited using the CATIA V5 application. In addition a reference to the original input file will be maintained in the parent CATProduct. This will allow the user to track any changes by using the CATIA V5 Update Status Checker command. Therefore enabling the derived representations to be updated when the referenced files are updated
- **Translator** – When selecting Translator mode any CATPart's that are created will be able to edited using standard CATIA V5 modelling methods. No links to the original input file are maintained. Therefore the user must handle any changes that may be necessary if the original input file changes.

From the **Others** section of the menu the user can control the creation of coordinate systems with the derived CGR files using the **Save Coorsys in Cgr** option. In addition the creation of FTA output is controlled by the **3D Annotation** option.

The default location of derived output data can be controlled by setting the required folder in the **Output Path** option. Note is this is going to be a common path for a group of users then you must ensure that the common folder is writable by all of the associated users.

Note! For the 3D Annotation selection to create FTA data there is a pre-requisite for the user to have a valid CATIA V5 FTA license as well as the optional Theorem PMI processing option license.

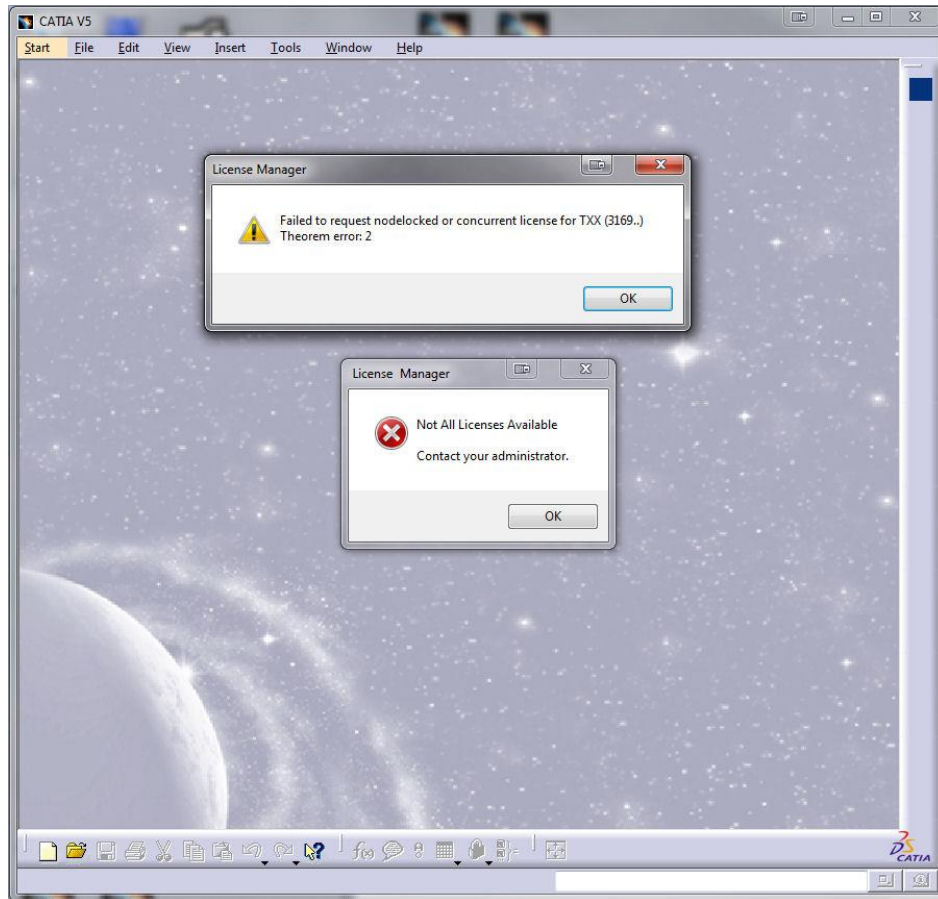
3. Ensure the *Document Environment* for **Folder** is set to state “Allowed” or “Current” in the Tools>General>Document panel. This will permit file selection for the “Insert Existing Component” command



Troubleshooting

CATIA Start Up Issues

If, when you start CATIA V5 you see the following messages:



You need to ensure that the Theorem FlexNet (FLEXlm) license server is running and that it is accessible within your network. This is usually achieved by setting the environment variable:

```
set THEOREM_LICENSE_FILE=7601@license_server_hostname
```

Where:

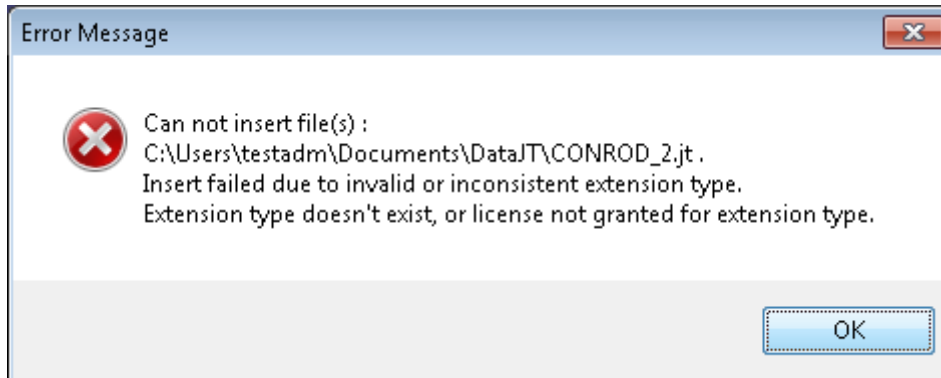
- The string `license_server_hostname` is replaced by the hostname of the computer that is assigned as the FlexNet license server machine or tri-server.

For a complete description of the FlexNet license server configuration and setup see the FlexNet Publisher documents supplied in the software installation directory named:

```
%TS_INST%\flexlm\doc\fnp_11_10_doclib_lf.pdf
```

Insert Existing Component Issues

If, when you attempt the command “Insert Existing Component” interactively in CATIA V5, you see an error message of this type:



This is caused because you have not allocated the TXX license in the CATIA V5 Tools>Options>General – Licensing page as specified in the Post Installation Set-Up section of this guide.

Maintenance Support

If you need any clarification of this information, please contact either your reseller product support organization or the Theorem Solutions regional support organization via email at:

For Europe and Asia Pacific region contact support@theorem.com

Or

For Americas region contact support-usa@theorem.com