All short reports – FreeCAD connector – TO BE MADE INTO PROPER DOCUMENTATION

For the field of computer-aided design (CAD), NVIDIA Omniverse presents an opportunity to improve on the workflows currently in place for both the initial design and continuous improvement of engineering components. Mainly, Omniverse serves as a shared 3D model exchange where contributors can create, view, and analyse the same model all while simultaneously conducting simulations in a true-to-life virtual environment [1].

The creation and analysis of engineering components in the context of Omniverse can be done using third-party connectors. Connectors offer real-time synchronisation between Omniverse and design and simulation tools currently in use. Due to Omniverse’s open platform, several Omniverse connectors have been developed such as for Unreal Engine, ParaView, and Autodesk Maya, with the potential of developing further connectors through the Connect library [2].

FreeCAD is a general-purpose 3D computer-aided design tool. While not necessarily the main industry standard for CAD, its’ open-source nature and modular architecture has allowed for introduction of numerous third-party ‘workbench’ tools expanding to a wider range of uses around engineering. For example, FreeCAD has previously been used to parse the nested structures and complicated surfaces associated with Monte-Carlo neutronics calculations for magnetic fusion devices [3].

Currently, NVIDIA does not offer a connector between Omniverse and FreeCAD. Fortunately, the development of such a connector is made straightforward due to the availability of both open Omniverse Connect and FreeCAD Workbench platforms.

## Current progress

The first version of a bi-directional connector between FreeCAD and Omniverse has been developed. The connector has the following features:

1. Git-style push and pull: This software supports existing USD files hosted on a Nucleus server. Users can upload to and download from a specified Omniverse Nucleus link.
2. Nucleus checkpointing: Git-style commit ‘checkpoints’ are automatically added to the file on Nucleus, with each checkpoint dated and identified by a unique token.
3. Multi-user support: The connector checks for file permissions and follows access rules set by the Nucleus administrator.

### FreeCAD connector user interface

The OmniverseConnector menu item on the FreeCAD toolbar has commands that can send and obtain CAD geometry from an NVIDIA Omniverse Nucleus.

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| **Menu items** | **Description** |
| Pull from Nucleus | Downloads a USD file from a specified Nucleus URL into a working document |
| Push to Nucleus | Uploads a selected working document to a specified Nucleus URL |
| Input USD Omniverse Nucleus URL | Opens a panel where user can specify a Nucleus URL and checks for URL validity and file permissions |
| Clear local junk files | Clears local junk files created by Connector. Must be called at the start and end of a working session. |

### FreeCAD connector usage

#### Setting up a USD connection

1. Select ‘Clear local junk files’ to reset any old connections which were previously made. Then, select ' Input USD Omniverse Nucleus URL’ to open the following panel:

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1. In the input panel, specify the long Nucleus URL. Ensure that this URL contains the path and filename of the file designated to be opened on FreeCAD. Click OK.
2. If the user is not logged in to a Nucleus server, clicking OK will trigger a login page on the user’s web browser. Enter the username and password like below:

A screenshot of a login screen

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1. If the username and password is valid and the user has valid file permissions, the following message will show in the report view. The user can now download and upload files to an existing USD file on the specified Nucleus.

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#### Downloading geometry from a specified Nucleus USD

1. Downloading geometry from a Nucleus specified in the previous section is done by clicking the ‘Pull from Nucleus’ button. Upon clicking the button, the report view will show the following, with a unique version identifier:

A screenshot of a computer

Description automatically generated with medium confidence

1. The user can now work with the downloaded geometry using the tools available on FreeCAD.

#### Uploading geometry to a specified Nucleus USD

1. Updating geometry to a Nucleus USD specified in the previous section is first done by selecting a FreeCAD Object from the Model tree. The Object is selected if a blue band is shown in its background:

A screenshot of a computer

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1. Once selected, the user can click the ‘Push to Nucleus’ button to update the USD file specified in the first section. Upon clicking the button, the report view will show the following, with a unique version identifier. The upload is also shown in the Nucleus Launcher app as a checkpoint on the selected file.

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### System process flow

#### Downloading geometry from a specified Nucleus USD

Prior to downloading geometry from Nucleus, the user must specify the URL of the designated USD file using the ‘Input USD Omniverse Nucleus URL’ button. Upon clicking this button, the system will check for the validity of the URL and access permissions of the user account. The ‘Pull from Nucleus’ button can then be selected. If permissions and URL are valid, a script is called which takes the USD file, converts it into an STL file, and saves it into a local temporary directory. Once done, the script calls a FreeCAD routine which imports the STL file into the user’s CAD workspace.

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#### Uploading geometry to a specified Nucleus USD

Much like the previous subsection, the user must have already inputted a URL of a valid USD file. When the user selects the ‘Push to Nucleus’ button, the system checks for URL validity and its file permissions. Additionally, it checks whether the user has selected a geometry from the Model tree. If all of these conditions are valid, a script is called which exports the selected FreeCAD geometry into an STL file in a temporary directory. A script outside of FreeCAD reads the STL file and converts it into a USD file which is then uploaded on to the Omniverse Nucleus.

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## Future work

For this work, the next steps mainly involve the improvement and addition of features to the FreeCAD software, addressing its current limitations such as:

1. Automatic prompts of geometry updates: There is no support for automatic checking of new checkpoints on Nucleus - so users will have to check for changes manually. This feature can make use of periodic API calls to Nucleus on latest checkpoints.
2. Backtracking to earlier checkpoints on FreeCAD: Currently, users can backtrack to checkpoints on Omniverse Create/View, but not on FreeCAD, unless the file HEAD is reverted to an earlier checkpoint.
3. Stages with multiple meshes: The current software only supports single-geometry USD files. For scenes with multiple mesh geometries, it is recommended to specify the referenced geometry USD file. Improvements can include a user interface for selecting geometry within FreeCAD.

## References

[1] A. Herrera, ‘The Metaverse: Separating the Wheat from the Chaff, from a CAD Perspective’, *Cadalyst*, Dec. 16, 2021. Accessed: Jun. 15, 2023. [Online]. Available: <https://www.cadalyst.com/collaboration/digital-twin/metaverse-separating-wheat-chaff-cad-perspective-78906>

[2] NVIDIA Corporation, ‘NVIDIA Connect Overview’, *NVIDIA Omniverse Documentation*, Jun. 14, 2023. https://docs.omniverse.nvidia.com/con\_connect/con\_connect/overview.html (accessed Jun. 15, 2023).

[3] J.-Y. Li *et al.*, ‘FreeCAD based Monte Carlo modeling approach for fusion reactor facilities’, *Fusion Engineering and Design*, vol. 155, p. 111711, Jun. 2020, doi: 10.1016/j.fusengdes.2020.111711.

# September Project Report - FreeCAD Connector

Raska Soemantoro

# Introduction

In the previous report, a prototype software of a bi-directional connector between NVIDIA Omniverse and FreeCAD was introduced. This report explains progress made on the software since the submission of the previous report.

Version 1 of the software relied heavily on the Pixar Universal Scene Description (USD) format. However, as geometry in the USD format is stored exclusively in tessellated form, this introduced a number of issues when editing the geometry within FreeCAD. Information that was initially attached to the CAD component in the FreeCAD workspace was not propagated to the USD file, as the file format conversion only considered the geometry of the component. This was because the transfer was achieved by firstly converting the CAD file (typically .STEP files) into a tessellated STL file, which was then converted into USD, as shown in Fig. 1(a). The method of pulling the geometry from the Nucleus was also similar as shown in Fig. 1(b), however this tessellated geometry had to be repaired prior to any editing, often requiring the user to manually re-define each CAD surface after fetching the file from the Nucleus storage server. Naturally, this meant that after multiple pulls and pushes between Omniverse and FreeCAD, the geometry would be tessellated and re-built many times, changing the geometry ever so slightly in each pass that the underlying geometry of the final model could be affected.

Aside from this major issue, the previous report outlined next steps, suggesting the addition of features to the developed software, addressing its limitations such as:

1. Automatic prompts of geometry updates: There is no support for automatic checking of new checkpoints on Nucleus - so users will have to check for changes manually. This feature can make use of periodic API calls to Nucleus on latest checkpoints.
2. Backtracking to earlier checkpoints on FreeCAD: Currently, users can backtrack to checkpoints on Omniverse Create/View, but not on FreeCAD, unless the file HEAD is reverted to an earlier checkpoint.
3. Stages with multiple meshes: The current software only supports single-geometry USD files. For scenes with multiple mesh geometries, it is recommended to specify the referenced geometry USD file. Improvements can include a user interface for selecting geometry within FreeCAD.

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| *A screenshot of a computer program  Description automatically generated*  *(a)* | *A screenshot of a computer  Description automatically generated with medium confidence*  *(b)* |
| *Fig. 1. (a) System flow of geometry (a) upload to and (b) download from Nucleus in version 1 of the FreeCAD-Omniverse Connector.* | |

# Method

The second version of a bi-directional connector between FreeCAD and Omniverse is currently in the development stage. In this version, a number of changes were made to address the limitations of the previous version and add further functionality to the software. This is broken down in the following sections.

## Changes

The main change in this version is that CAD geometry is no longer just stored as USD files. In this change, geometry is pushed to the Nucleus in a neutral .STP format, which preserves the information initially attached by FreeCAD and does not require tessellation of the geometry, thus no longer requiring the user to repair the geometry of the downloaded file. This is shown in Fig. 2. Along with the STP file, the connector still uploads a USD file, maintaining the benefits of using the USD file format. The uploaded USD file is updated whenever a new version of its corresponding STP file is made. However, the connector now imports only the stored STP file when pulling from Nucleus. In version 2 of the connector, the tessellation process is done only once for each version of the geometry, in place of the practically unlimited layers of earlier geometry approximations that can be done in version 1.

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| *Fig.2. Simplified upload and download workflow of FreeCAD connector version 2.* |

Another change is the user workflow prior to downloading from and uploading geometry to the Nucleus. In the previous version, users had to input the specific link of their USD geometry. FreeCAD then imports the file before writing back to the USD once changes are pushed. In the current version, USD and STP files are stored in a project folder, which contains single-component assets and assemblies. By default, projects are stored in omniverse://$HOST\_NAME/Users/$USER\_NAME/FreeCAD/$PROJECT\_NAME. Assets are then stored in $PROJECT\_FOLDER/asset\_$ASSETNAME/ and assemblies stored as $PROJECT\_FOLDER/assembly/$ASSEMBLY\_NAME.usda. This is shown in Fig 3.

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| *Fig. 3. Representation of the default file storage tree in FreeCAD connector version 2.* |

Establishing a connection with a component stored on the Nucleus is now a project-based process: the user can either open an existing project or create a new project, and then select the asset they require. When opening an existing project, clicking the ‘Open existing project’ button will prompt the user to input the Nucleus link of their project. Similarly, clicking the ‘Create new project’ button will prompt the user to input the Nucleus hostname and the new project name. Once done, clicking the ‘Create/browse project assets’ button opens a pop-up menu prompting the user to select an existing asset or create a new one. Once the STP and USD asset URLs are available to view on the panel, the user can push and pull geometry to and from the Nucleus in one click. Pictorial representation of the UI is shown in Fig. 4.

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| *A screenshot of a computer  Description automatically generated*  *(b)* | *A screenshot of a computer error  Description automatically generated*  *(c)* |
| *A screenshot of a computer  Description automatically generated*  *(d)* | |
| *Fig. 4. Omniverse Nucleus connection settings UI (a) main page, (b) upon clicking ‘create new project’, (c) upon clicking ‘open existing project’, and (d) upon clicking ‘create/browse project assets’* | |

Some minor changes were made to the software. For example, the user is now able to access the component’s Omniverse properties on the FreeCAD GUI. This can be viewed by clicking the object, which shows its properties just under the model tree. These properties are attached to the FreeCAD objects and include the last sync time with Nucleus, the USD and STP links, and the last version ID of the component. Of particular interest is the Nucleus USD and STP links, which are used on occasion to locate the component on Nucleus. In the previous version, links are stored in a temporary directory, which can be a concern when dealing with multiple files. This version also checks for links stored as a FreeCAD property. If there is a conflict between the link stored in the temporary file and as a FreeCAD property, the connector throws an error and requests the user to select the correct component link. This is shown in Fig. 5.

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| *A screenshot of a computer  Description automatically generated*  *(a)* | *A screenshot of a computer  Description automatically generated*  *(b)* |
| *Fig. 5. (a) viewing Omniverse properties of a component in the FreeCAD UI and (b) example of a property conflict requiring manual intervention of the user.* | |

## In development: assembly tools

Currently, the assembly tools feature of the FreeCAD connector is still in development. This tool is aimed to allow users to interact with the assembly functionalities of various workbenches on FreeCAD such as that found on the standard Part workbench, [A2plus](https://wiki.freecad.org/A2plus_Workbench), [Assembly3](https://wiki.freecad.org/Assembly3_Workbench), or [Assembly4](https://wiki.freecad.org/Assembly4_Workbench), and propagate the component placement to a USD file on the Omniverse Nucleus.

Some main software features have been developed. In the assembly tools panel of the FreeCAD connector, two options are now available: create new assembly and open existing assembly. The first option pulls up a pop-up menu, prompting the user to input an assembly name and to select items in the FreeCAD workspace to include in the assembly. Note that these items are only ones that have already been pushed to the Nucleus. The user will have to first push each item manually before they can make an assembly out of the desired items. Upon clicking ‘create new assembly’, a new USDA file will be made in $PROJECT\_FOLDER/assembly/, containing only references to the USD component, that can be visualised on Omniverse in the same placement as that on FreeCAD. Meanwhile, clicking ‘open existing assembly’ on the assembly panel will prompt the user to select an existing USD file to import. The entire assembly will then be imported into the FreeCAD workspace. An example image of the FreeCAD and Omniverse UI showing the same multi-component assembly is shown in Fig. 6.

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| *A screenshot of a computer  Description automatically generated*  *(b)* |
| *Fig. 5. FreeCAD connector assembly tools (a) main panel, (b) upon clicking ‘open existing assembly’, and (c) upon clicking ‘create new assembly’.* | |

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| A screenshot of a computer  Description automatically generated |
| *Fig. 6. FreeCAD connector version 2 UI showing a sample multi-component assembly.* |

## Future work

Next steps of the work mainly revolve around refining the single-asset push and pull process, as well as adding more features to the assembly tool. The assembly tool at its current state is still envisioned as a git-style push and pull system, however if improvements can be made to speed up the time it takes to request data from Nucleus, a live-link would be possible. This can be done by extracting the Xform positions of each component in a USD live layer and converting them into FreeCAD object placement coordinates.

**December Project Report - FreeCAD Connector**

Raska Soemantoro

In this report, an explanation of the progress made on the FreeCAD Connector since the submission of the previous report is given.

Version 1 of the FreeCAD connector relied heavily on tessellated Universal Scene Description (USD) geometry. The second version of the software introduced the use of neutral CAD files in the STEP format and the ability to work with multiple components in a git-style push and pull workflow. The shift from USD to STEP in the second version was important as the first version required the user to repair the geometry due to the tessellated format at every step of the editing process, which is not only a lengthy manual process, but also introduced errors in the geometry as it is tessellated and re-built continuously.

This version mainly focused on the multi-component assembly feature of the developed software, which includes performance improvements in the push-pull assembly workflow and the introduction of an interactive live assembly feature.

# Push-pull assembly workflow

The push-pull assembly workflow provides a way for the FreeCAD user to use the various assembly functionalities of FreeCAD such as that found on the standard Part workbench, [A2plus](https://wiki.freecad.org/A2plus_Workbench), [Assembly3](https://wiki.freecad.org/Assembly3_Workbench), or [Assembly4](https://wiki.freecad.org/Assembly4_Workbench), and propagate the component placement to a USD file on the Omniverse Nucleus.

The main user interface for the assembly feature in the FreeCAD connector is the Assembly Tools panel. A screenshot of this is given inFig. 1. There are two main UI areas in this panel, which in total contains 5 different buttons. The Create New Assembly button allows the user to create a USDA file on the Nucleus server which references the USD objects in the current FreeCAD workspace. Clicking this button opens a pop-up menu that prompts the user to input an assembly name and to select items in the workspace to include in the assembly. The Import Existing Assembly button will open a pop-up menu that prompts the user to select an existing assembly from the current project folder. Upon selection, the entire assembly and its components will be imported into the FreeCAD workspace. These two features were introduced in version 2 of the FreeCAD connector, and detailed in the previous report.

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| *Fig. 1. FreeCAD Connector Assembly Tools panel with a project and assembly loaded.* |

Under these two buttons, status indicators are given, detailing the current project and assembly in use in the given session. If both current project and assembly elements are valid, the status will show the ‘Ready’ indicator, which allows users to access the Upload assembly changes, Fetch assembly changes, and Live assembly mode features.

The remaining three buttons allow the user to transfer positional information (translation and rotation) to and from the Nucleus server. The first two are part of the batch push-and-pull assembly workflow. The Upload assembly changes button sends the cartesian coordinates and rotation of each component in the current FreeCAD assembly to the assembly USDA file stored on the Nucleus and alters the position of each referenced single component within the assembly USDA file. Meanwhile, the Fetch new assembly changes button requests the coordinates and rotation of each element in the assembly USDA file stored in the Nucleus server, and adjusts each component’s rotation and translation in the FreeCAD workspace. These two buttons allow for simple manipulation of position and angle of the components, integrated with the Omniverse environment.

# Live assembly mode

An exciting addition to the Omniverse Connector for FreeCAD is the Live assembly mode. This button allows for live real-time communication between the FreeCAD workspace and Omniverse environment. Upon clicking this button, a pop-up menu will prompt the user to select an available Omniverse Live session attached to the assembly USDA file. These available sessions are also possible to view on the Omniverse USD Composer app. This is shown in Figs. 2 (a) and (b).

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| A screenshot of a computer  Description automatically generated  *(a)* | A screenshot of a computer  Description automatically generated  *(b)* |
| *Fig. 2. Available Live Sessions pop-up dialog on (a) FreeCAD and (b) Omniverse USD Composer.* | |

On FreeCAD, the Live Assembly Mode button will show that it is active and toggled in blue, as shown in Fig 3(a). If the assembly USDA file is opened on Omniverse USD Composer and a user is logged into the same Live Session as the FreeCAD user, a small user icon will appear near the Live button of the USD Composer app. Also, a message will appear on the USD Composer notifying the user that another user has joined the session. These are shown in Figs. 3(b) and ©. Any changes done on the Omniverse side will now propagate in real-time on FreeCAD. Clicking the Live assembly button again on FreeCAD deactivates the Live mode, triggering the Omniverse Client to quit the Live Session and thus turning the button white in the process.

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| A screenshot of a computer screen  Description automatically generated  *(a)* | A screen shot of a computer  Description automatically generated  *(b)* |
| A blue rectangle with black text  Description automatically generated  *(c)* | |
| *Fig.3. Indications that Live Session is active: (a) Toggled Live assembly button active on FreeCAD, (b) User icon next to Live button on USD Composer, and (c) User joining message on USD Composer.* | |