

# **SolidWorks API Project Manager: Application Manual**

CCET 4610 Spring 2023

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I. Introduction

In order to start the application on a school computer please see the accompanying document “Starting\_Final\_Application.pdf” found both in this directory and with the client program.

II. Application Layout



A| The connection status area

B| New Project Button

C| Edit Project Button (not implemented)

D| Update Button

E| Application and API Settings Button

F| Project Tree

G| File Tree

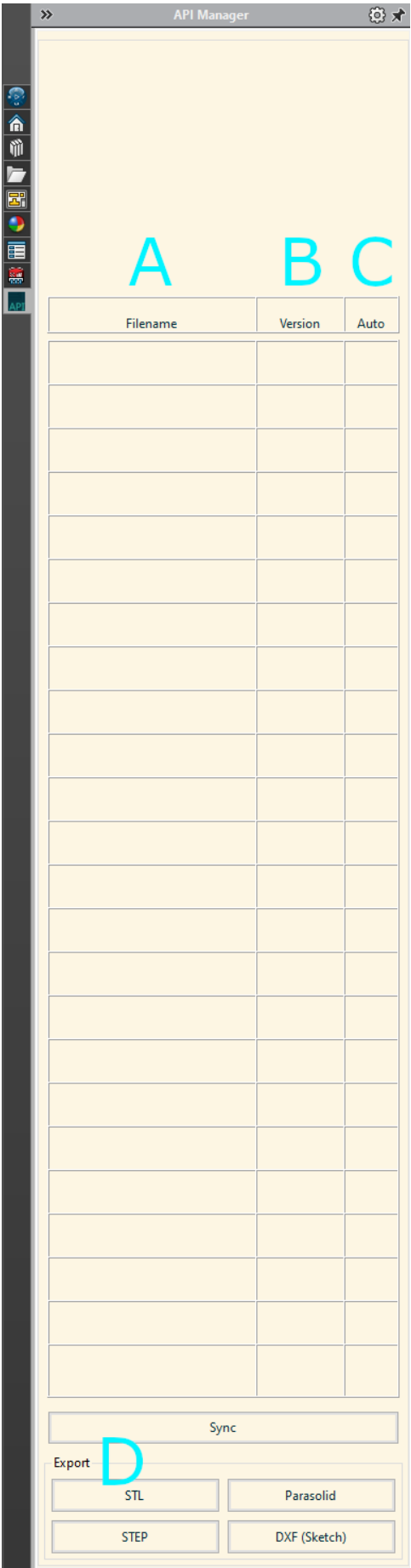
H| New File Button

I| Active Project

J| Selected File

K| New File Button

III. Taskpane Layout



A| Open File Name Column

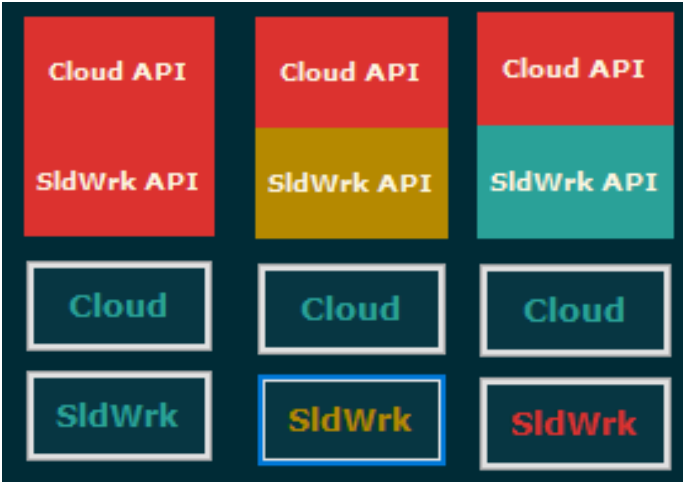
B| Open File Version Column

C| Open File Sync Column

D| Quick-Export Buttons

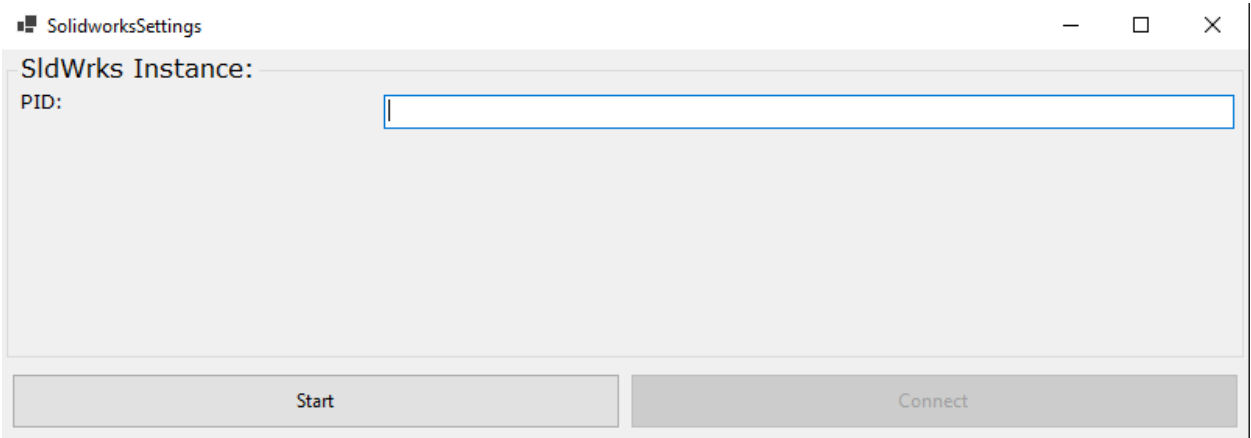
IV. Connections and Settings

Solidworks Connection:



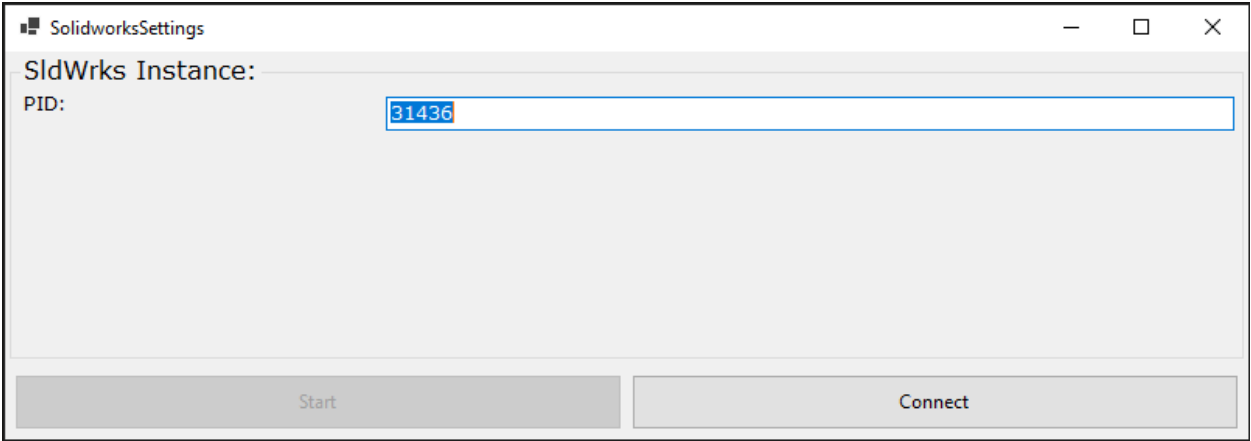
The lower box labeled “SldWrk API” shows the status of the Solidworks instance connection. When showing red the instance is disconnected and when cyan it is connected. The orange status color is used when a new instance of Solidworks is launched and the process ID has already been grabbed by the application.

Pressing the SldWrk button will show the SolidworksSettings dialog box.



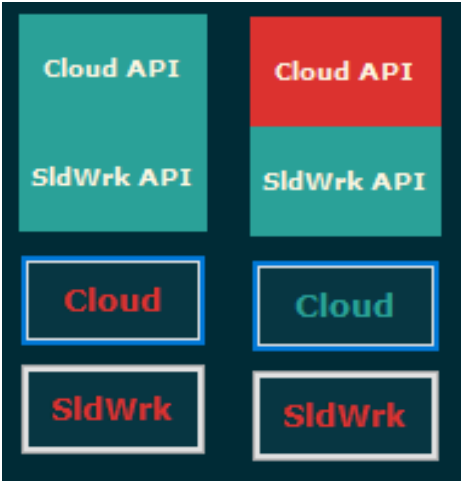
When opening on a personal computer the process ID can easily be found with the task manager, by going to the details tab and finding the SLDWRK.exe process. This can be entered and the Connect button pressed. Upon correctly entering the process ID the status box will turn cyan.

The school and other domains don’t have this kind of functionality and the Start button will need to be pressed. The dialog will disappear, and the status box will turn orange while Solidworks launches in the foreground. Only after Solidworks has fully launched the SldWrk button can be pressed again and you will be presented with the same dialog box, only with the PID filled out this time.

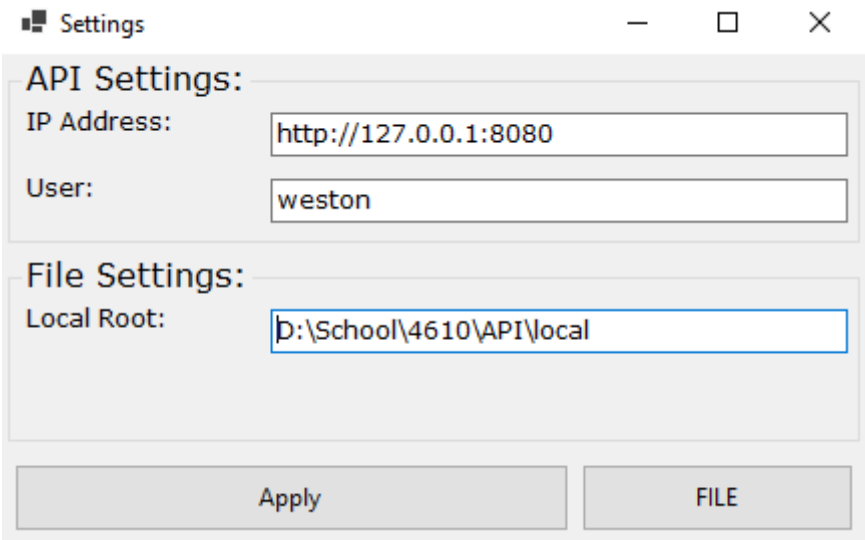


Press the Connect button and check for the cyan colored status box if waiting for the Solidworks instance to start. Once the connection has been made the API Manager taskpane will show up in Solidworks. The taskpane may need to be resized in order to refresh the sizing.

API Connection:



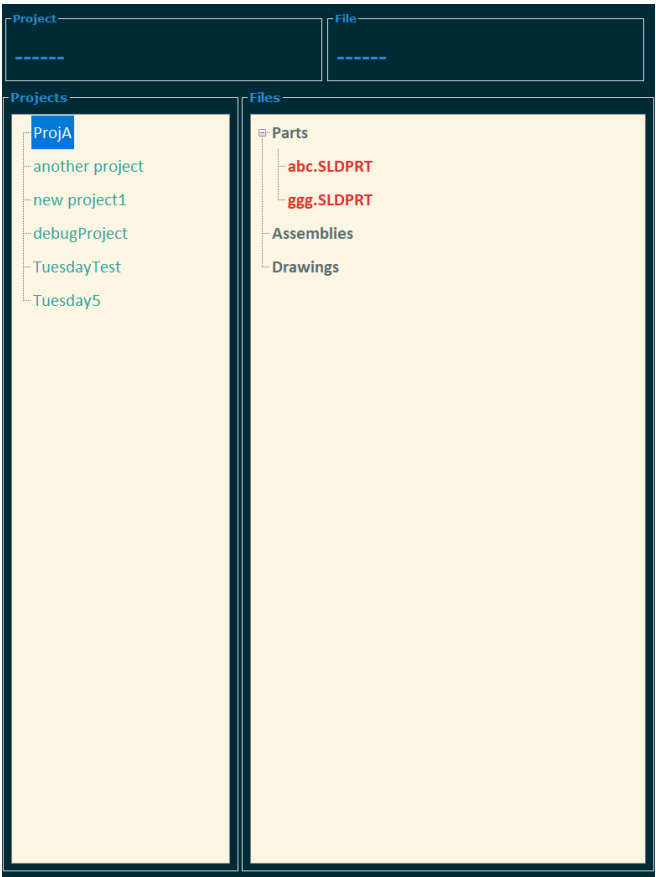
The upper box, labeled “Cloud API” shows the connections status to the cloud server. If red the connection has failed, if cyan the application has connected to the server.



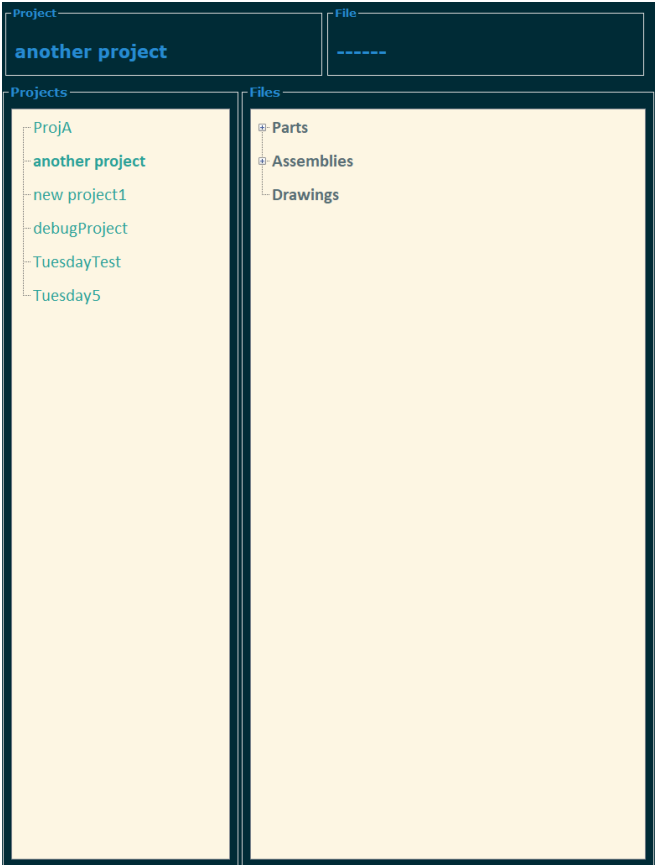
In order to connect to the server, select the Settings Button in the lower left-hand corner to bring up the Settings dialog. Both the IP Address and User boxes are filled out, making sure to include the last forward slash in the address. Upon clicking the local root box and Choose Folder dialog box will appear and the head directory for local file storage should be selected. This will automatically populate the box with the path and the Apply button can be selected.

Optionally the FILE button can be used to load settings from a file instead of filling out the form, this is more of a debugging option but also has practical end use.

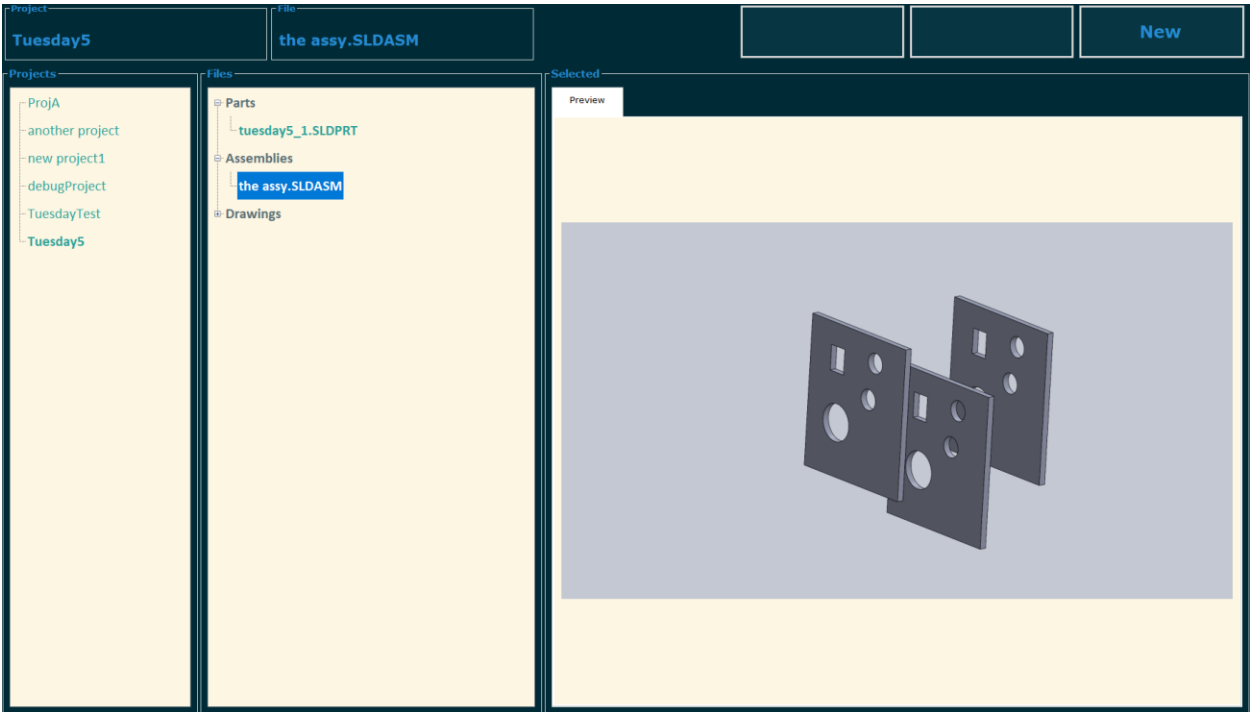
V. Project and File Trees



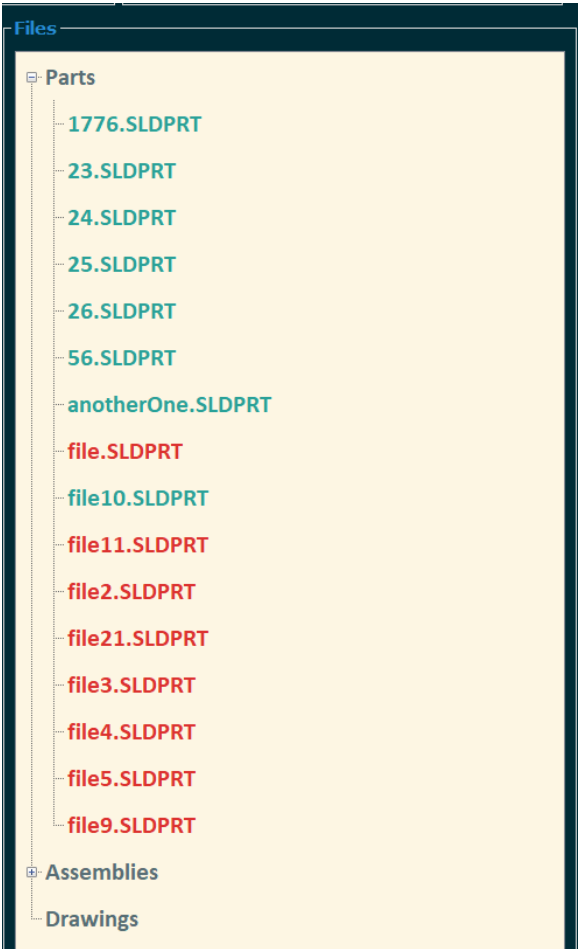
The Projects tree can be navigated by either single clicking on the items or double clicking. A single click will change the files showing in the Files Tree without modifying the Active Project. When double clicking on a tree item the Project label will change notifying the user that file actions will be on that project.



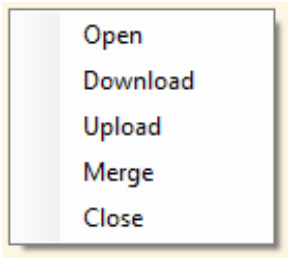
The File Tree can be used to navigate the current files in a project. The files are sorted depending on the type of Solidwors file and the below image shows the image preview that comes from single clicking on an item.



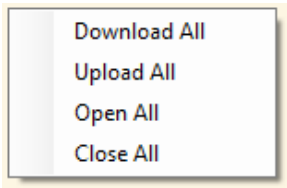
VI. Local vs. Cloud Files



The items in the File Tree will show either red to indicate the local version either doesn't exist or is outdated or cyan if the file is up to date with the server.

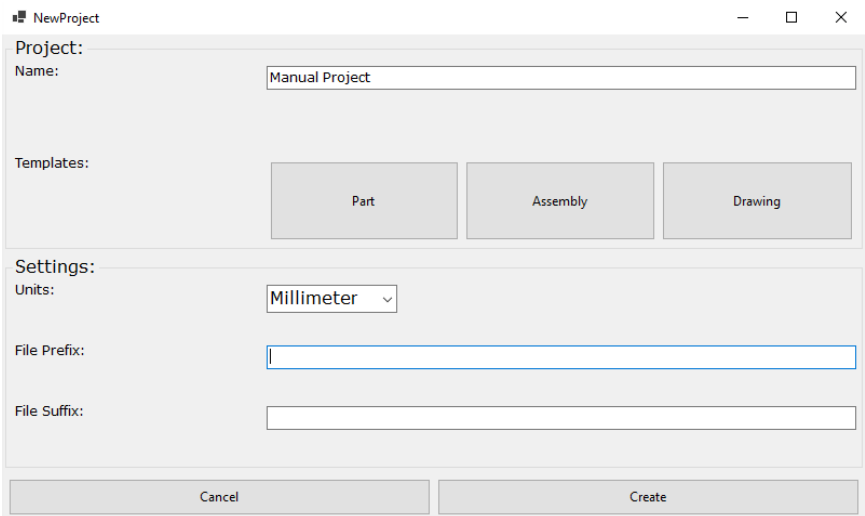


After selecting a file and right clicking somewhere inside of the File Tree the context menu will show. Selecting open will open the file if it exists locally or will prompt if the file needs to be downloaded. Selecting download will only download the file from the server. The merge and close buttons are both for future implementation.

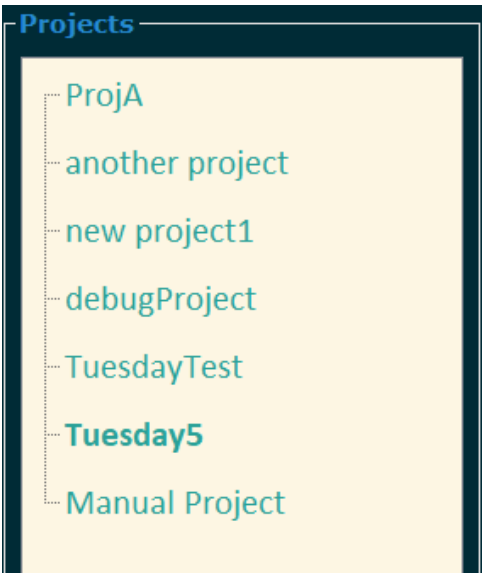


Much like the File Tree context menu, after double clicking a project to activate it you can right click on the Project Tree to open its context menu. This menu is similar to the above where only the Download and Open functions work right now. They will both recursively open and download all the files in a project.

VII. Creating a New Project



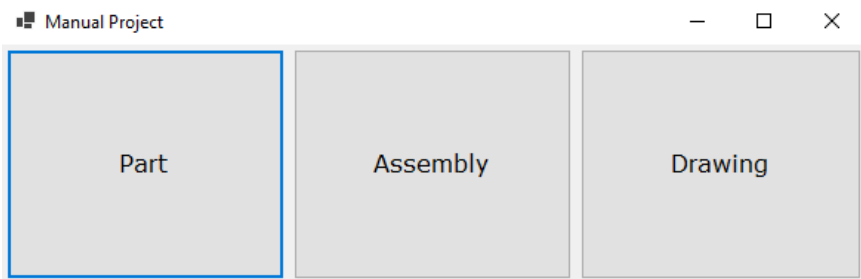
With or without a project selected you can select the New Project button to launch the NewProject dialog. The only aspect of this dialog that works currently is the name for the project, the rest of the data is sent to the server by the application but is not used yet. I eventually want to have the ability to specify specific templates per project. After filling out the name the Create button can be pressed.



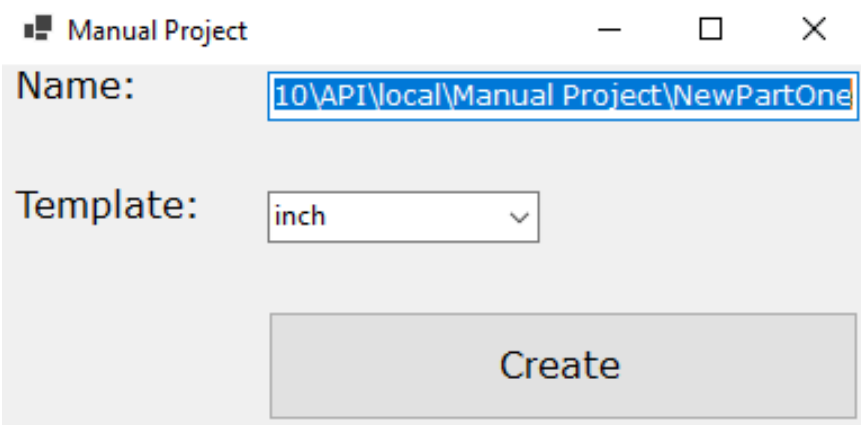
Looking at the Projects tree now the new Project has been created.



VIII. Creating a New File



Before pressing the New button it’s important to make sure a project has been activated by double clicking it. Then New and select the type of part that will be created. Upon selected the type of part a window will show prompting you to select a folder path and a filename. The dialog will open to the root folder for the project, be sure to only store the file inside that folder or a sub folder of that folder. The extension does not need to be added.



After the file path has been selected you will be prompted for type of template. Eventually there will be the functionality for other templates but selected one of the built in ones will modify the units for you automatically upon creation.

After selecting a template, click the Create button and observe the Solidworks window open the file for you. This new file will also be added to the Open File table in the taskpane similar to the below image. This signals that the file’s events are being actively watched by the application now.

Filename	Version	Auto
NewPartOne.SLDPRT	0	

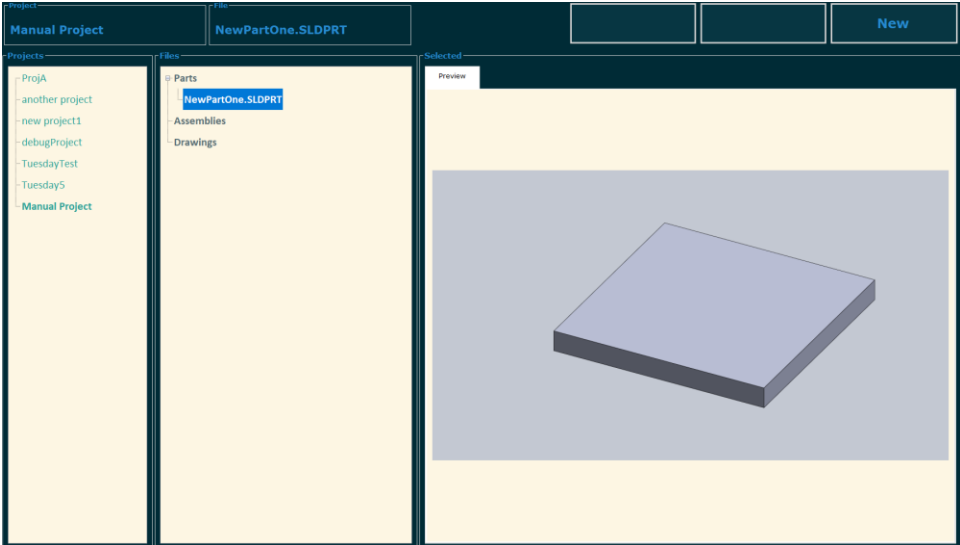
IX. Taskpane & Versions

Filename	Version	Auto
NewPartOne.SLDPRT	1	

Create a simple extrusion in this new file for demonstration purposes and save the file. The application will listen for the successful save event and will launch its own series of syncing processes outlined in more detail in the Documentation. Simply put, the view is fit to the screen and the preview image is captured. Then the saved file is copied to a temporary location and encoded into the string format to send to the server.

The version for each file will display it’s own status color depending on the sync status of the file. When cyan the cloud version is up to date with the last local save and when orange the file is newer on the disk.

Finding the newly created file in the tree you can see that the preview has been created or updated for the file.



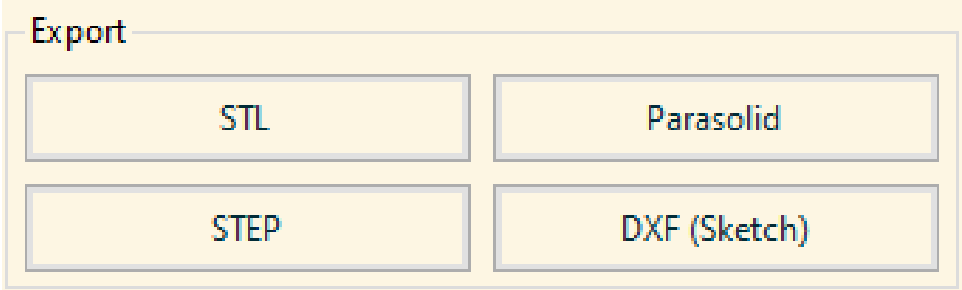
Filename	Version	Auto
NewPartOne.SLDPRT	1	

The auto button for each file will toggle the automatic sync on every file save. When this isn’t cyan the application will do nothing upon saving the file, which can be helpful for conserving data. When toggling the setting back on the next save will result in a file sync to the server.

Filename	Version	Auto
NewPartOne.SLDPRT	2	

Pressing the filename for one of the Open Files in the taskpane will automatically switch the view to that file. This functionality can save a large amount of time designing large assemblies with lots of parts and I hope to eventually allow color coding of each entry.

**X. Taskpane Quick-Export Buttons**



Pressing The STL, Parasolid, and STEP buttons will quickly export the active documents as that type of file with a timestamp as the suffix. These filetype are the most common to export to and this option allows for a quick workflow when exporting to something like a printing slicer.

The DXF (Sketch) button works much the same way and is a little misleading in its' title. I am planning on prompting as to whether the user wants to export as dxf or dwg but currently only the dwg export works. This needs to have a face of the body selected before clicking and will export that face as a dwg file with a similar timestamp suffix.