CAD File Transfer and Backups

In engineering and mechanical design, the need to exchange CAD data is required on a routine basis. Parametric CAD systems maintain links between files that can fail if the files are missing or if their locations have changed. This can cause a seemingly simple exchange of CAD data to become problematic when the CAD files are opened by the recipient.
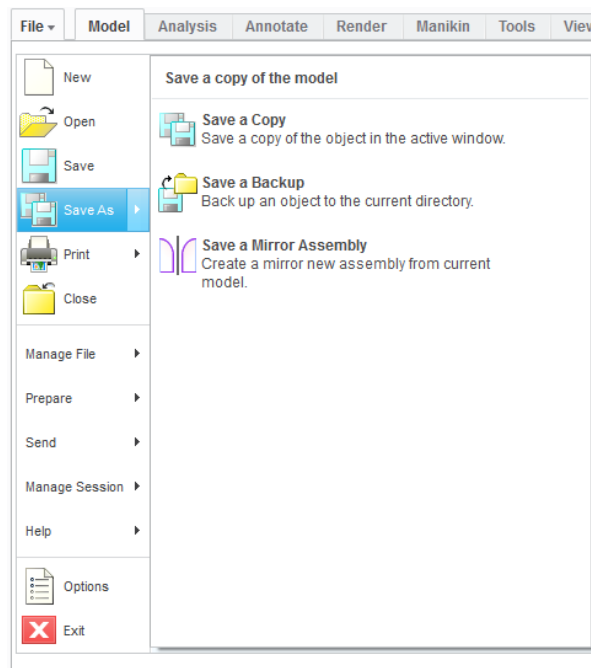
To address this issue most CAD systems include provisions to prevent such errors when preparing models for backup or transfer. The instructions below describe the process for two of the most common CAD platforms, Creo (ProEngineer) and SolidWorks.

Creo (ProEngineer)

Saving out the required CAD files in Creo is done by choosing "Save a Backup". This tool automatically packages all referenced files into the new file location. This ensures that all accompanying files are included as part of the backup. If the model contains suppressed components, ensure they are in session before performing the backup or else they will not be included.

A useful tip when performing a backup with Creo is to backup the drawing of the part or assembly. This ensures the drawing will be included. Each drawing will need to be opened and backed up individually.

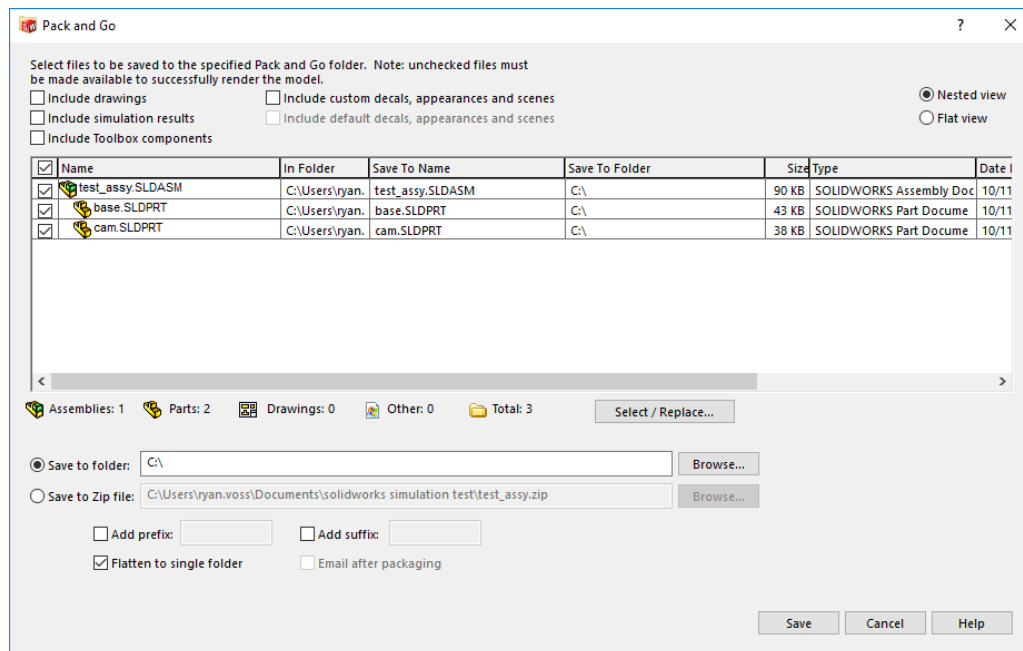Locate "Save a Backup" by going to **File>Save As>Save a Backup.**

Menu for locating the "Save a Backup" option.

SolidWorks

SolidWorks creatively titles their backup option "Pack and Go". This tool automatically finds all of the referenced documents and saves them to a folder or a zip file. This makes it easy to ensure that all necessary files are transferred.

There are 3 methods for activating the SolidWorks Pack and Go tool:
1. Open the upper level file in SolidWorks and go to **File** > **Pack and Go.**
   - You can start from either the top level assembly or accompanying drawing, and all parts or subassemblies included will be packaged.
     - Make sure to check the "include drawings" box to have all part drawings packaged also.
2. Right-click the file in Windows Explorer, select **SolidWorks** > **Pack and Go.**
3. In SolidWorks Explorer, select the file and click the **Pack and Go** Icon.
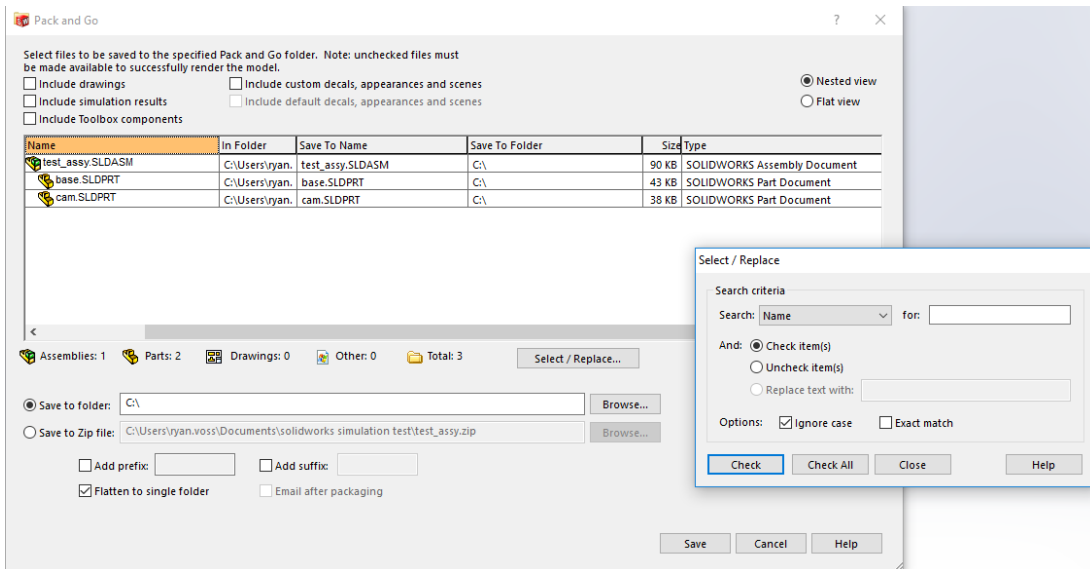

Pack and Go dialog box

Options:
- All referenced documents are listed in a nested view similar to the design tree. You have the option to change to a flat view.
- The check boxes beside the file names can be deselected to exclude specific files.
- Option to save all files to a folder or automatically compress into a zip file.
- **"Flatten to a single folder"** will throw all files in the main directory that you are saving to. Unselecting this option will recreate your current folder structure within the new folder.
- Ability to add a prefix or suffix to saved out files (Shown under **"Save To Name"** column).
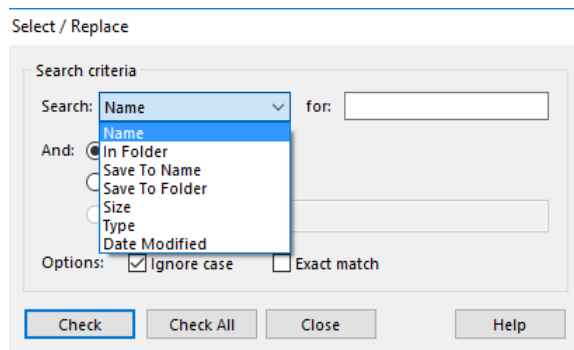- Selecting **"Include drawings"** will also grab all drawing files associated with the selected assemblies and parts.

- Choose **"Include simulation results"** or **"Include decals, appearances and scenes"** as required.

Advanced Options:
- **"Name"** – removes specific filenames from being included using **"Uncheck"**.
- **"In Folder"** – if your toolbox files are stored in the standard "SolidWorks Data" folder search this and uncheck so you don't need to send these files every time.
- **"Save to Name"** – use **"Replace text with"** to modify the job number in all of the files.
- **"Save to Folder"** – use **"Replace text with"** to modify the folder names that it will be saved to.
- **"Type"** – uncheck all assembly and drawing files so only the part files are saved.
- **"Date Modified"** – include only items modified on a specific date to avoid large file transfers.

Select/Replace dialog box with the available advanced options.

Drop-down menu for selecting different search criteria columns.