# Setting up Drawing Template File Locations

**Objective**: Know where to place UI ME drawing template files for easiest access.

**Instructions**: Download the *UI_ME_Templates.zip* file from Day 6 of the course Mindworks page and extract or copy the contents to a known folder (SolidWorks will not be able to access files contained in a zipped folder). Then, follow one of the next two sections to setup file paths in SolidWorks. If you have administrator rights on your computer, follow the section called *Setup with Administrator Rights.* Otherwise, you can follow the Quick Setup approach.
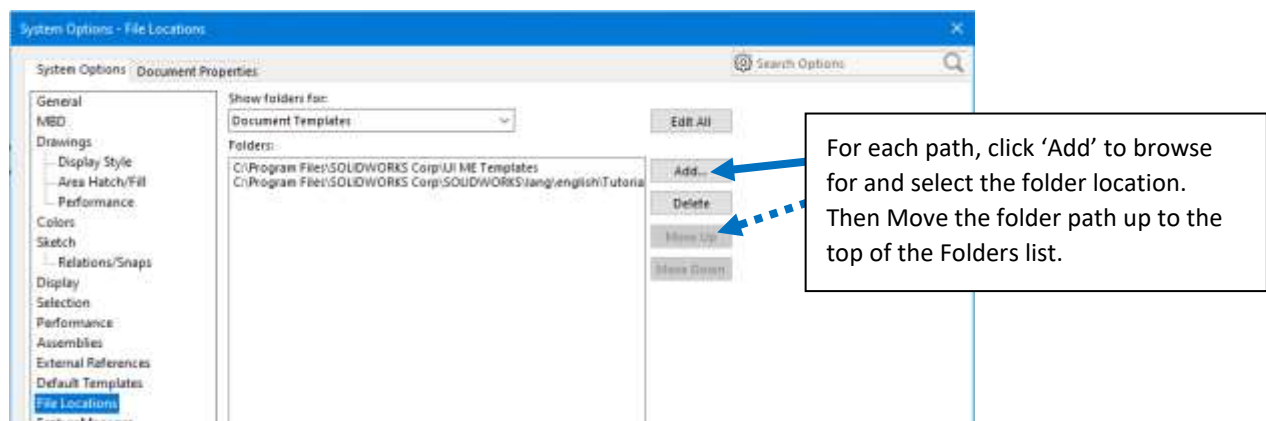


Figure 1. Downloaded and extracted files from Mindworks Day 6.

## Quick Setup

The following steps will allow you to access the templates in SolidWorks but will require you to browse for your new drawing templates folder when setting up a new drawing. Alternatively, if you have administrator rights on your pc, you can skip to the next section on *Setup with Administrator Rights*.

If you don't have administrator rights on the pc you are using (such as for on-campus computers), you can save the entire folder to a local or shared drive location and add the file path to SolidWorks via:

1. Tools → Options → System Options → File Locations → Show folders for: Document Templates
2. Tools → Options → System Options → File Locations → Show folders for: BOM Templates
3. Tools → Options → System Options → File Locations → Show folders for: Revision Table Templates



For each path, click 'Add' to browse for and select the folder location. Then Move the folder path up to the top of the Folders list.

The next steps are only available if you have administrator rights on the CAD machine you are using.

## Setup with Administrator Rights

To take advantage of the folders that SolidWorks will use by default (this will be more efficient in the long run), you can place each type of file in the location where SolidWorks will naturally look for it. This will allow you to more seamlessly use and access the customized templates.

**Solidworks Part (.prtdot), Assembly (.asmdot), Drawing (.drwdot) Template Files**

These are the files that solidworks references when you select File → New. SolidWorks looks for these at: **C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\lang\english\Tutorial**

**Solidworks Drawing Sheet Format (.slddrt) Template Files**

These are the files that solidworks references when you select File → New. SolidWorks looks for these at: **C:\ProgramData\SOLIDWORKS\SOLIDWORKS 2020\lang\english\sheetformat**

**BOM Table (.sldbomtbt) and Revision Table (.sldrevtbt) Template Files**

SolidWorks looks for these files at: **C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\lang\english**

**Note:** the downloaded templates are missing the part *Quantity* information. You will need to add this in to the part file properties Custom tab and resave the file as a Part Template file (.prtdot). You will also need to insert a column to the BOM table and resave the file as a BOM template file (.sldbomtbt). Save a BOM template by right-clicking on the BOM table and selecting 'Save As'.