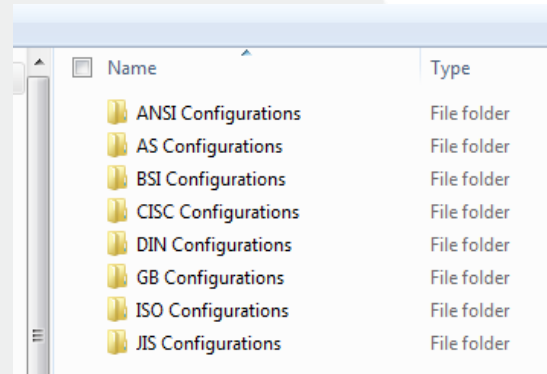# SOLIDWORKS Configured Weldment Profiles

Objective

This tutorial is to show the user how to download SOLIDWORKS configured weldment profiles and configure SolidWorks to apply the new profiles.
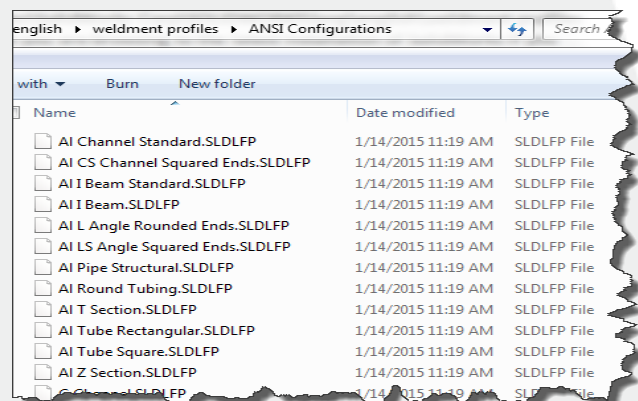
Where to Find It

- Please select http://blogs.solidworks.com/tech/wp-content/uploads/sites/4/Configured-Weldment-Profiles.zip (you may need to hold down control and click the link)
- This will download the Zip File with all of the Configured Weldment Profiles inside.
- Copy the contents of the 'Weldment Profiles' folder.

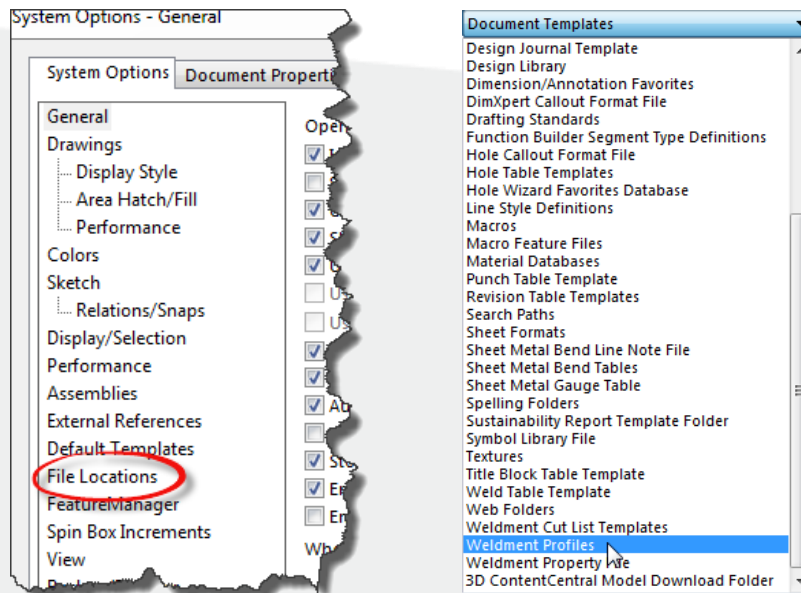## How to Configure SolidWorks for the Configured Weldment Profiles

Windows Explorer:

- Browse to C:\Program Files\SolidWorks Corp\SOLIDWORKS\lang\english\weldment profiles
  - Please ensure that you are browsing to the latest installation of SolidWorks if you have more than one installation (2014, 2015, etc.)
- Verify that there are profiles in each folder to ensure a complete download.
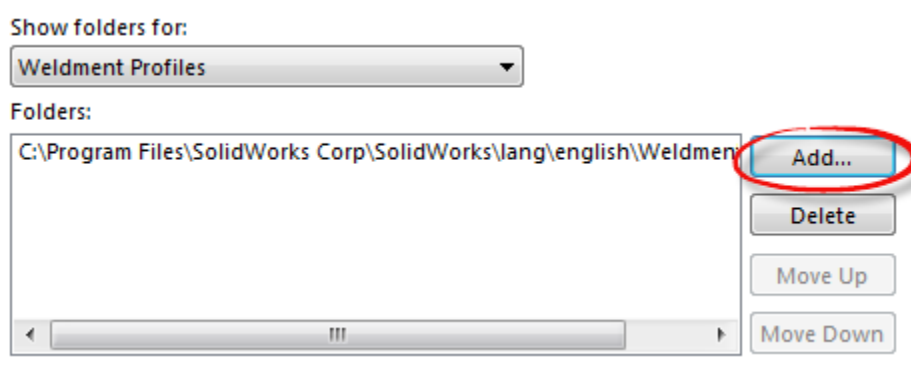
SOLIDWORKS:

- With SolidWorks Open, go to Tools>Options>System Options>File Locations
- Once in File Locations, select the drop down menu to select Weldment Profiles

- Add a file location for weldment Profiles

- Browse to the location where you pasted the Configured Profiles in windows explorer:
  C:\Program Files\SolidWorks Corp\SOLIDWORKS\lang\english\weldment profiles
  - Location may vary depending on your installation location
- Close Options

- Open/Create Part that you wish to apply the structural member to that will use the new Configured Weldment Profiles
- Start the Structural Member Command
- Select Standard (ie. ANSI Configurations)
- Select the type of Steel Member
- SOLIDWORKS now sees the size configurations in the same library feature part.
  - Before 2014 each weldment profile was a new library feature part file.

- Now when you want to make a change/add a size, you go into the steel type and add the configuration instead of creating a new file

- Thank You for viewing this tutorial from GoEngineer

- Enjoy!