

Installing Commonly Used Weldment Profiles

You can install standard weldment profiles for use in your models. The “ISO” and “Ansi Inch” folders are good weldment profiles to set up. Here, we show the setup for the “Unistrut” folder of weldment profiles.

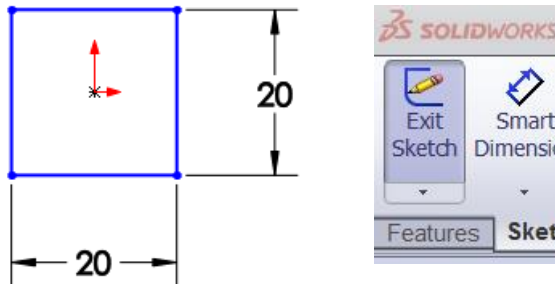
1. Go to “Design Library” on the right side of the SolidWorks screen and click “SolidWorks Content” and then “Weldments”
2. Next, hold down [Ctrl] and then click one of the compressed folders to install a group of weldment profiles. Here, [Ctrl] click on the “Unistrut” icon and save this zipped folder to “c:\temp\” or another location where you have write access (we do this because sometimes your computer won’t let you save directly to a subfolder under “c:\Program Files”
3. Navigate to “c:\temp\” and double click on the “Unistrut.zip” folder to see the zipped files.
4. Copy the unzipped “Unistrut” folder to the following location.

c:\Program Files\SolidWorks Corp\SolidWorks\lang\english\weldment profiles

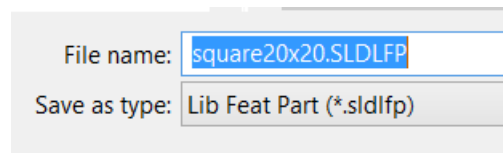
5. Your computer may ask for administrator permission to perform this copy.
6. These profiles should now show up when you use “Insert>Weldments>Structural Member” to associate a member (a line in a sketch) with a weldment profile.

Creating Your Own Weldment Profile

1. Draw the cross section in Solidworks and be sure to click “Exit Sketch” before saving the profile.
 - a. Create a sketch.



2. Give the sketch a name, and save it as a “Lib Feat Part (*.sldlfp)” to a folder where you have write access. That is, in some cases, your operating system may not allow you to write to the location in “Program Files” where SolidWorks source files are saved. In this case, just save the file to a temporary location on your hard drive (or you can change write permissions to the “weldment profiles” folder). **Make sure the part is highlighted in the “tree” when saving the “*.sldlfp” part file.**



3. Copy the file to the location in “Program Files” where SolidWorks cross sections are saved.

c:\Program Files\SolidWorks Corp\SolidWorks\lang\english\weldment profiles\ansi inch

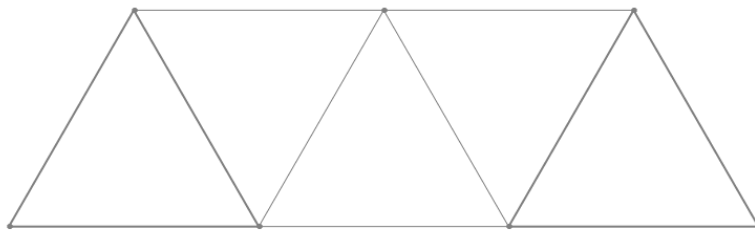
OR

c:\Program Files\SolidWorks Corp\SolidWorks\lang\english\weldment profiles\iso

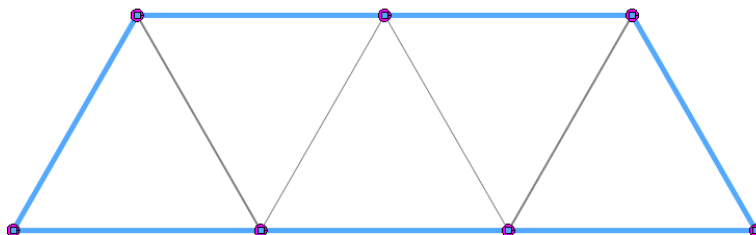
Since our dimensions are in millimeters, we will save the profile to a folder under “iso.” We can create our own folders under the “iso” folder to keep track of the profiles we create, or we can save the profile in an existing folder. Here, we save the profile under in a folder named “square tube.”

Using a Weldment Profile

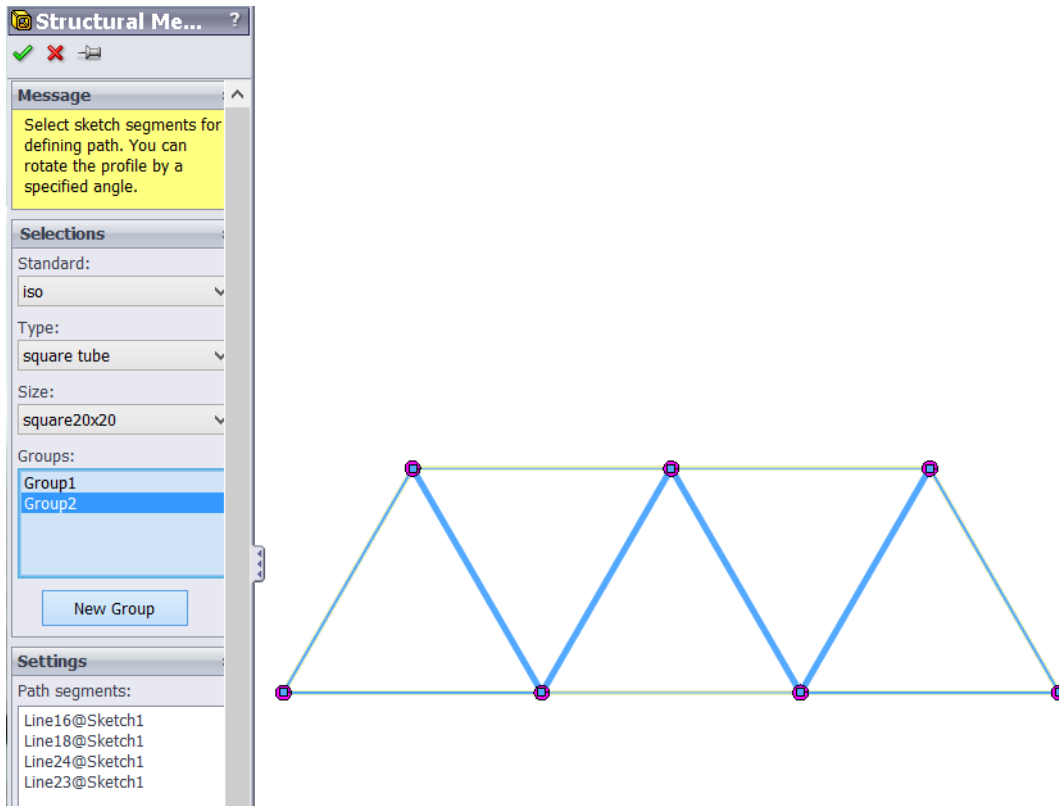
1. Draw the shape of your structure and save it as a part file. You only need to create a 2D sketch, not a 3D solid geometry. Be sure to click “Exit Sketch” before trying to define weldments.



2. Choose “Insert > Weldments > Structural Member” and then find the cross section you want. To use the one defined above, make the following selection:
 - a. Standard: iso
 - b. Type: square tube
 - c. Size: square20x20
3. Click the members that will use each size of Weldment. You will probably need to make a “New Group” for internal members.



The screen will look as shown below when defining the 2nd group of internal members. Note that we can use multiple member sizes and profiles in a model.



4. Your structure is now defined, and you can go into "Simulation" to set the material, member type (truss or beam), displacement boundary conditions, and loads.