

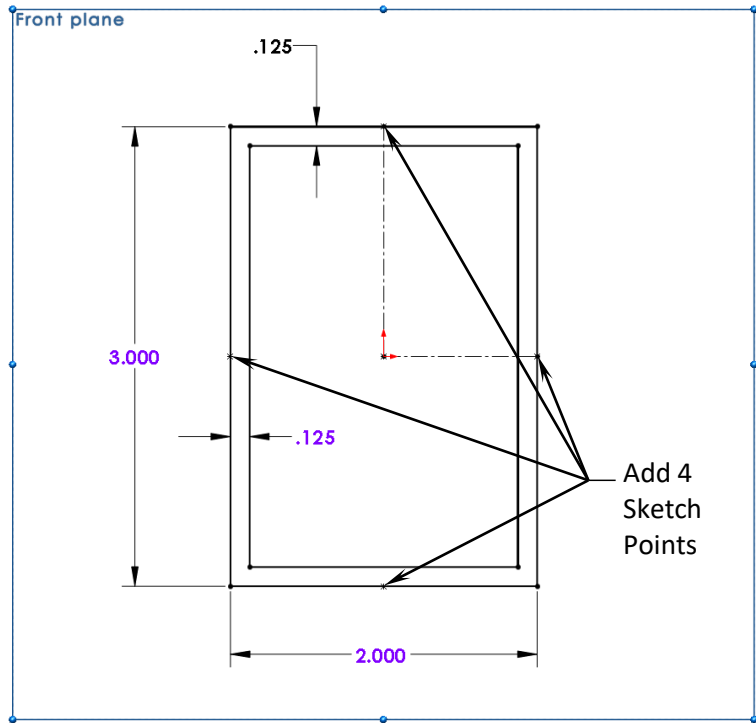
3. Sketching the 1st weldment profile:

Select the Front plane and open a **new sketch**.

Sketch **2 Rectangles** and add the dimensions as indicated to fully define the sketch.

Add the **Symmetric** relations where needed to reduce some of the redundant dimensions.

Add the **Sketch Point** as noted to help locating the profile later on.



4. Saving the profile:

Exit the Sketch. **Select Sketch1** from the Feature tree and click **File, Save As:** Browse to the following directory:

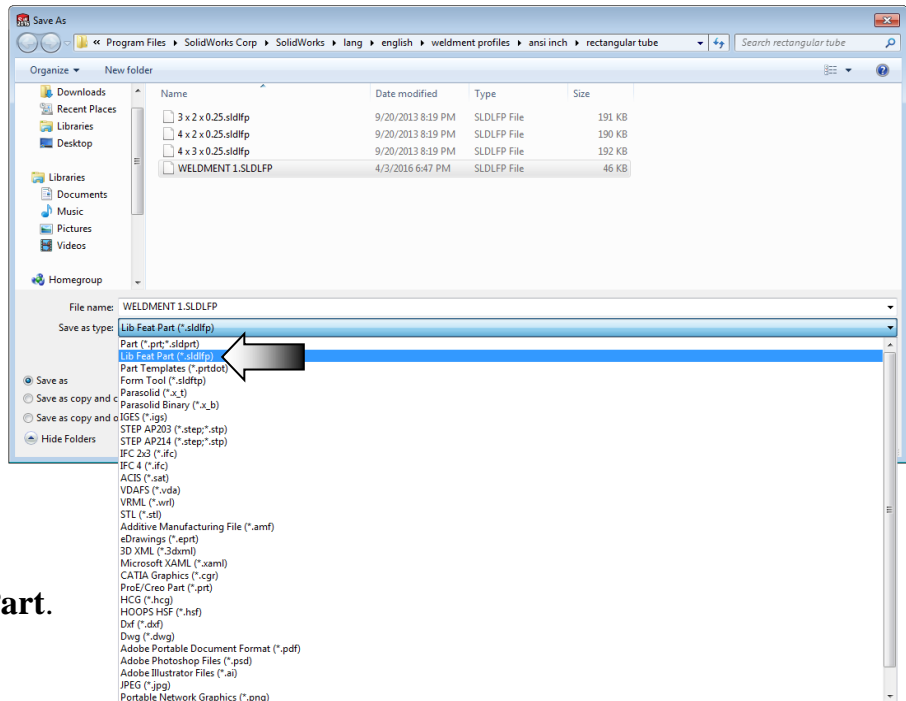
C: Program Files/SolidWorks Corp/SolidWorks/Lang/English/Weldment Profiles/Ansi Inch/Rectangular Tube.

For file name, enter:

WELDMENT 1.

For the file type, select **Lib Feat Part**.

Press **Save**.



5. Creating the 2nd weldment profile:

Click **File / New / Part**.

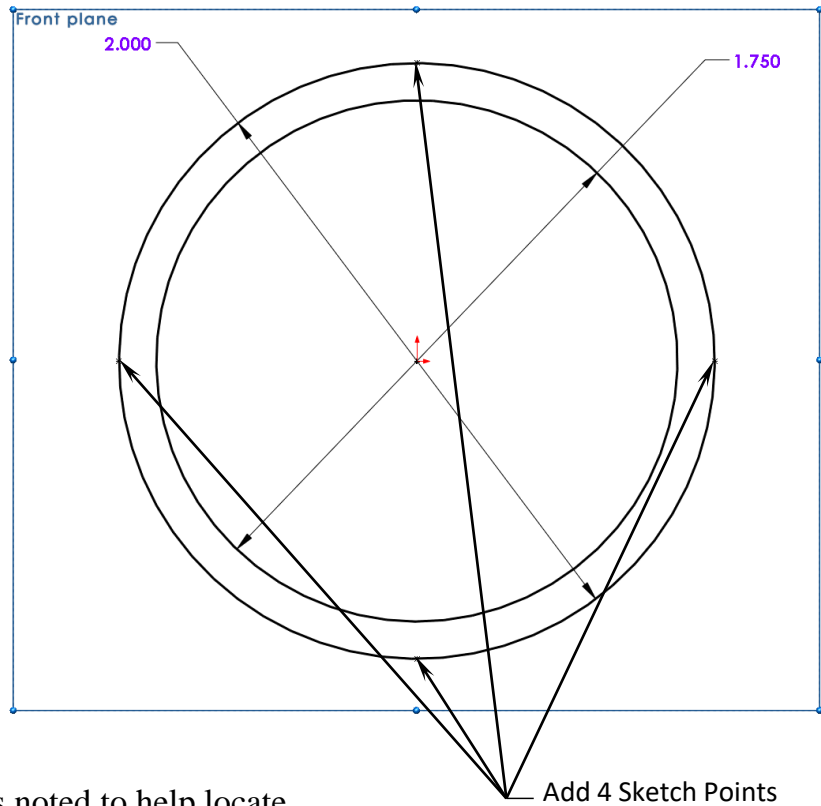
Select the default **Part Template** and set the Units to **Inches, 3 decimal places**.

6. Sketching the profile:

Open a **new sketch** on the **Front plane**.

Sketch **2 circles** and add the diameter dimensions to fully define the sketch.

Add **4 Sketch Points** as noted to help locate the profile later on.



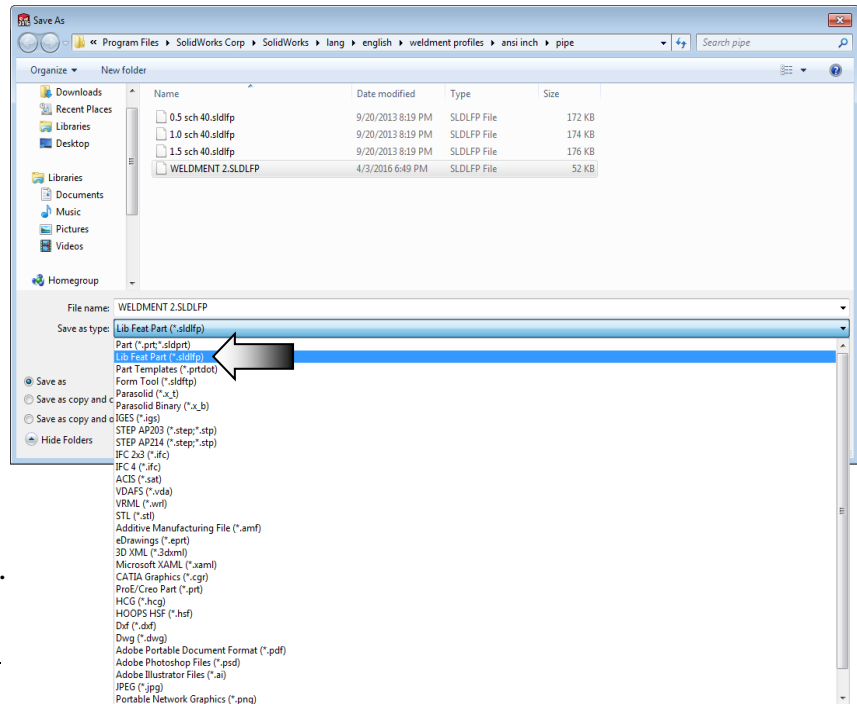
7. Saving the profile:

Exit the sketch. **Select Sketch1** and **save** it in the same directory as the last one.

For the file name, enter: **WELDMENT 2.**

For the file type, select **Lib Feat Part**.

Press **Save** and **close** all documents.

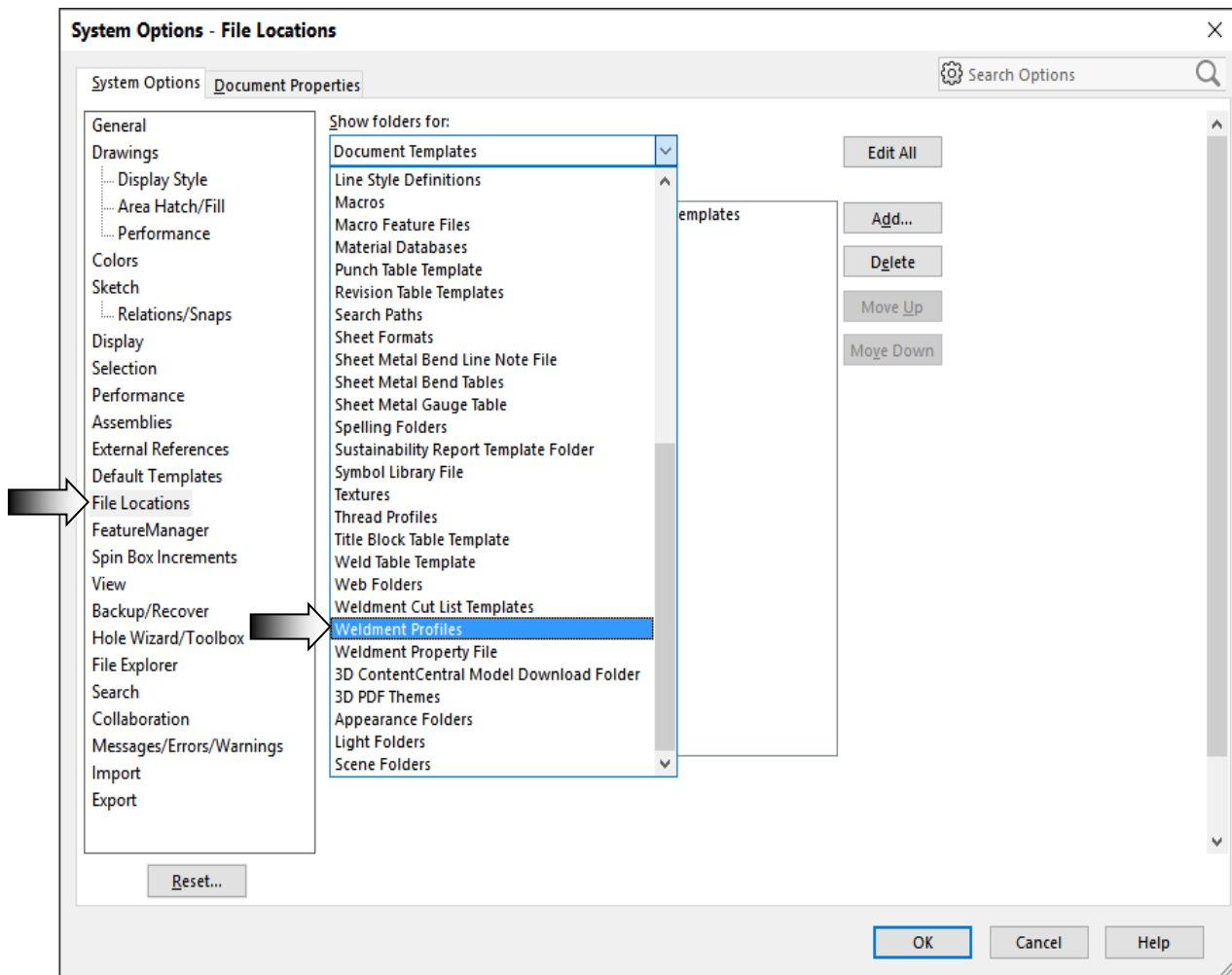


8. Showing the Weldment Profiles Folder:

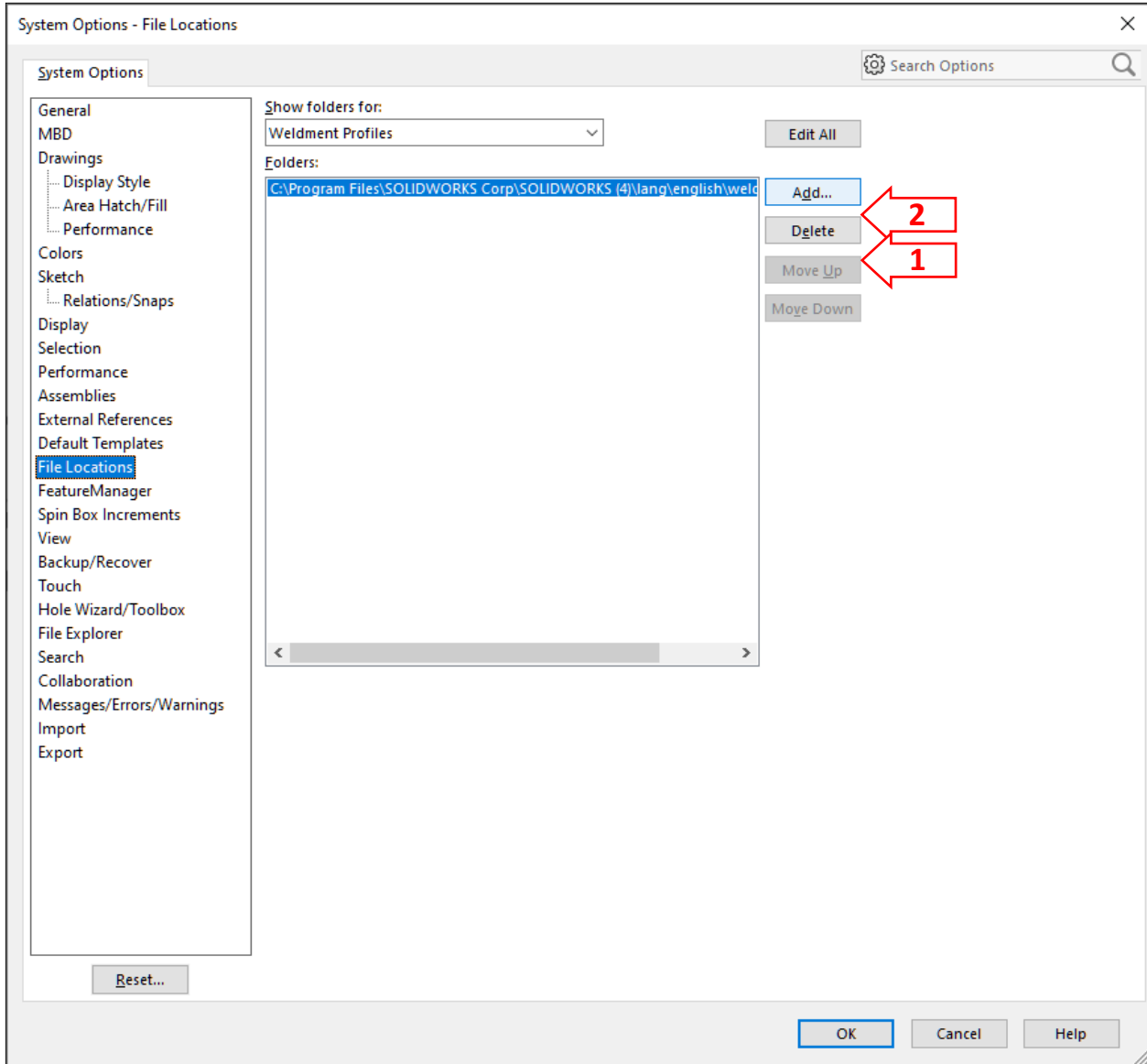
The option “Show Folder For” lists file types for which you can define a search path. You specify the folders to search for different types of document. Folders are searched in the order in which they are listed.

The Weldment Profiles are saved in the following directories: **C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\Lang\English\Weldment Profiles**.

Select **Options / File Locations / Show Folders For / Weldment Profiles** (arrows). If the Weldment Profiles folder is not visible, click **Add** and browse to the Weldment Profiles directory listed above.



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(You may need to be the Administrator to access this folder)

- 1. Delete** the existing location to the Weldment Profile folder.
- 2. Click Add** and browse to the location where you saved the Weldment Profiles. Select the **Weldment Profiles folder** and click **Select Folder**, then click **OK**.

If the option above failed:

- Create the Weldment Profiles as indicated in step 3 & 5 but save the profiles on the Desktop.
- Use Windows Explorer to copy and paste the weldment profiles from the Desktop to the Weldment Folder, then continue with step 9.