

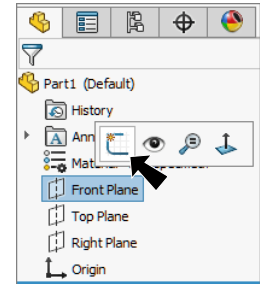


**A. Sketch 1.5 by 3 Rectangle.**


Step 1. Click File Menu > New, click **Part Metric** and OK.

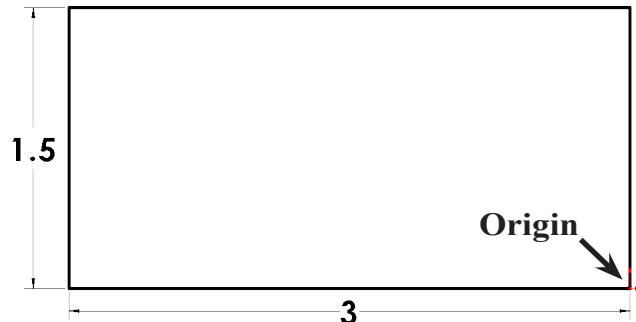
Step 2. Click **Front Plane**  in the Feature Manager and click **Sketch**  on the context toolbar, **Fig. 1**.




**Fig. 1**

Step 3. Click **Corner Rectangle**  (S) on the Sketch toolbar.

Step 4. Sketch a rectangle starting at the Origin , **Fig. 2**. The Origin should be in the bottom right corner of rectangle. This will make adding structural members a little easier.



**Fig. 2**


Step 5. Click **Smart Dimension**  (S) on the Sketch toolbar.

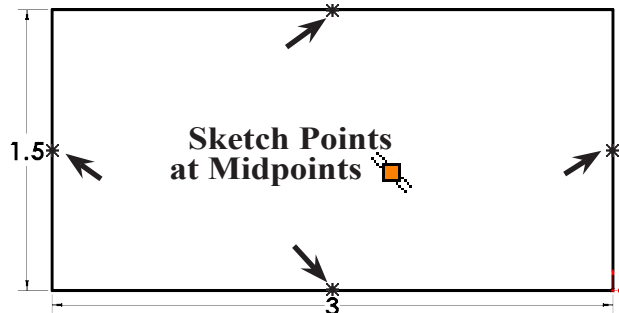
Step 6. Add dimensions, **Fig. 3**. 1.5 x 3

Step 7. Click **Zoom to Fit**  (F) on the View toolbar.

**B. Points.**

Step 1. Click **Point**  on the Sketch toolbar.

Step 2. Sketch Point at midpoint of each line of rectangle, **Fig. 3**. To find midpoint, move cursor across line and as cursor approaches midpoint highlights  you click.



**Fig. 3**

Step 3. Click **Exit Sketch**  on the Sketch toolbar.

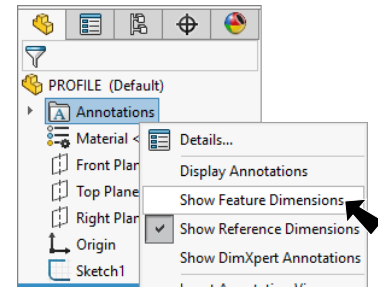
**C. Save as "PROFILE".**

Step 1. Click File Menu > Save As.

Step 2. Key-in **PROFILE** for the filename and press ENTER.

## D. Properties 1.5 by 3.

Step 1. **Right click Annotations**  in the Feature Manager and click **Show Feature Dimensions** from menu, **Fig. 4**.



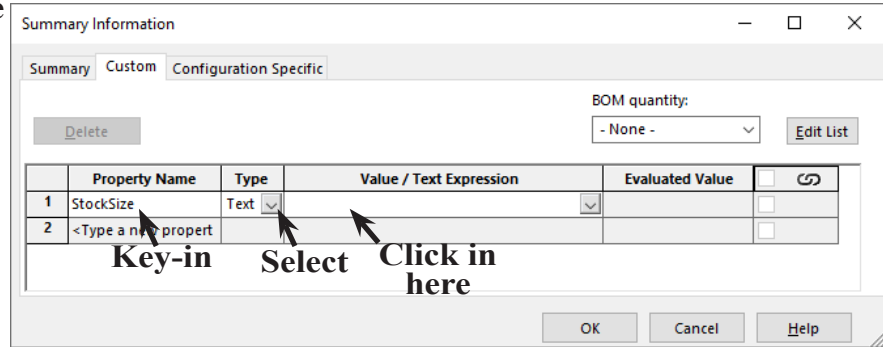
**Fig. 4**

Step 2. Click File Menu > Properties.

Step 3. In the Summary Information dialog box on the Custom tab set:

under Property Name, **Fig. 5**  
key-in **StockSize**  
under Type  
select **Text**

under Value/Text  
Expression  
click in box to  
select the field



**Key-in** **Select** **Click in here**

click **1.5 dimension**  
**in sketch, Fig. 6**. You might have to  
move Summary Information dialog  
out of the way.

**key-in space (spacebar) X space**  
**(spacebar), Fig. 7**

click **3 dimension** in sketch, **Fig. 8**

under Property  
Name (below  
StockSize), **Fig. 9**

key-in  
**Description**

under Type  
select **Text**  
under Value/Text  
Expression

key-in **1.5mm x 3mm**

press Tab key and **Evaluated Value** updates to:

**1.5 x 3**  
**1.5mm x 3mm**

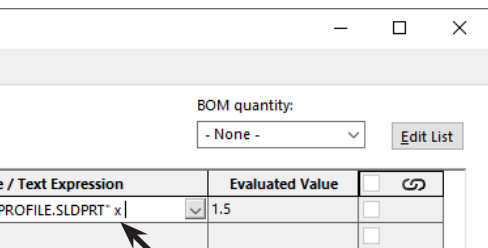
click **OK**.

Step 4. Save. Use **Ctrl-S**.

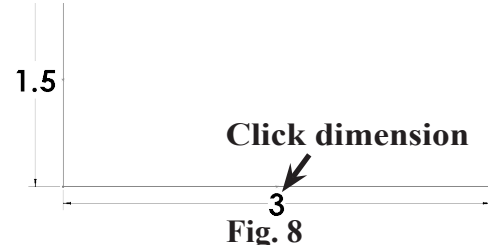
**Fig. 5**



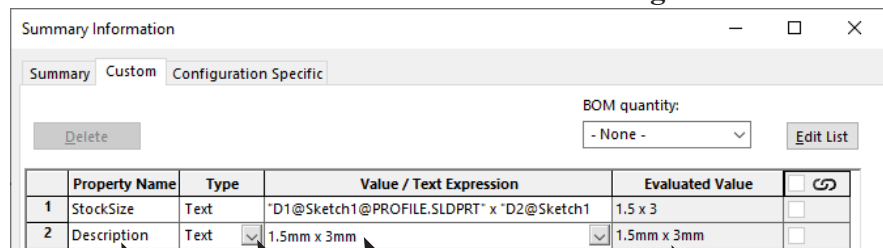
**Fig. 6**



**Fig. 7**



**Fig. 8**

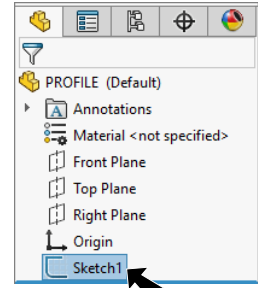


**Fig. 9**

**Key-in** **Select** **Key-in**

## E. Save as "1.5 X 3" Library Feature Part in Sub-folders.

Step 1. Click **Sketch1** in the Feature Manager to select the sketch, **Fig. 10**.  
The Sketch must be selected when you save as Lib Feat.



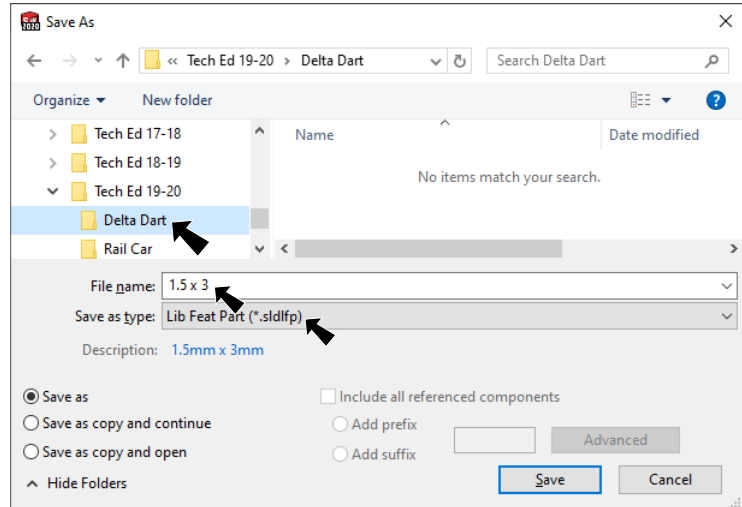
**Fig. 10**

Step 2. Click File Menu > Save As.

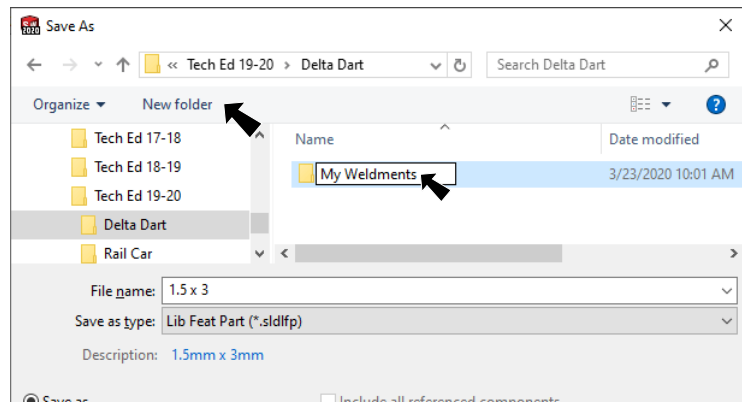
Step 3. In the Save As dialog box:  
key-in **1.5 x 3** for file name, **Fig. 11**  
set **Save as Type** to **Lib Feat Part**  
navigate to **My Documents/Tech Ed 19-20**

Step 4. Click **New Folder** button,  
**Fig. 12**  
key-in **My Weldments** for  
folder name  
**double click My Weldments**

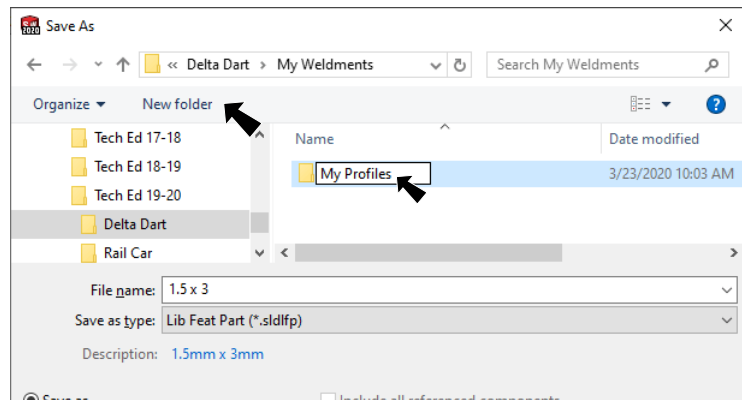
Step 5. Click **New Folder** button,  
**Fig. 13**  
key-in **My Profiles** for folder name  
**double click My Profiles**



**Fig. 11**



**Fig. 12**



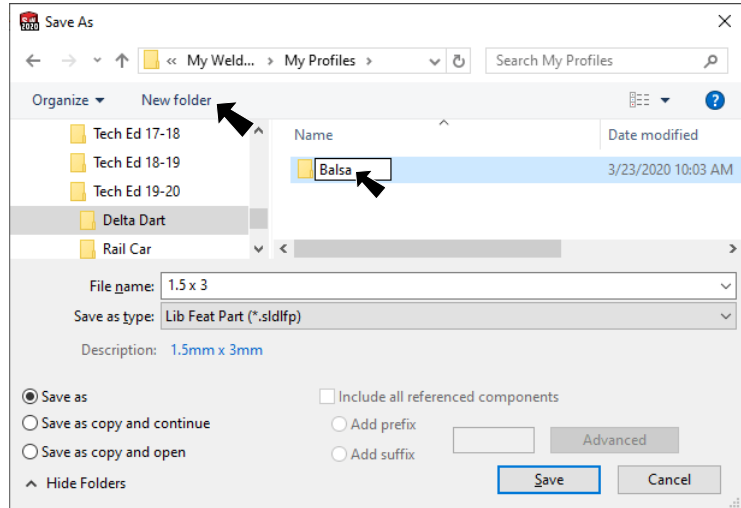
**Fig. 13**



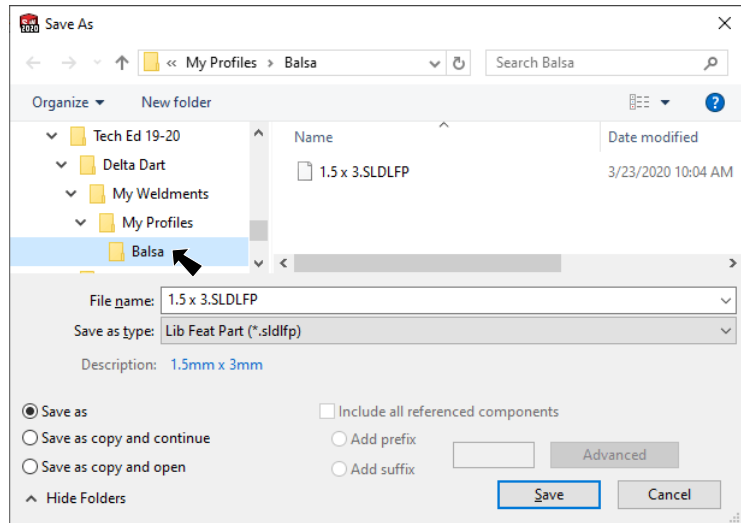
Step 6. Click **New Folder** button, **Fig. 14**  
key-in **Balsa** for folder name  
**double click Balsa folder**

Step 7. Confirm folders:  
We just created 3 subfolders,  
**Fig. 15**  
**My Weldments**  
**My Profiles**  
**Balsa**

click **Save**.



**Fig. 14**



**Fig. 15**

## F. Custom Profile File Location.

Step 1. Click **Options**  on the Standard toolbar.

Step 2. In the System Options dialog box,  
select **File Location** on left, **Fig. 16**  
under **Show folders for:**  
select **Weldment Profiles**

Step 3. If you **DO NOT** have  
permissions/rights to  
your SOLIDWORKS  
install folder continue  
here:

Click **Add** button,  
**Fig. 16.**  
Navigate to your **My**  
**Weldments** folder,  
**Fig. 17.**  
Click **Select Folder.**  
Click **OK.**  
Click **OK.**

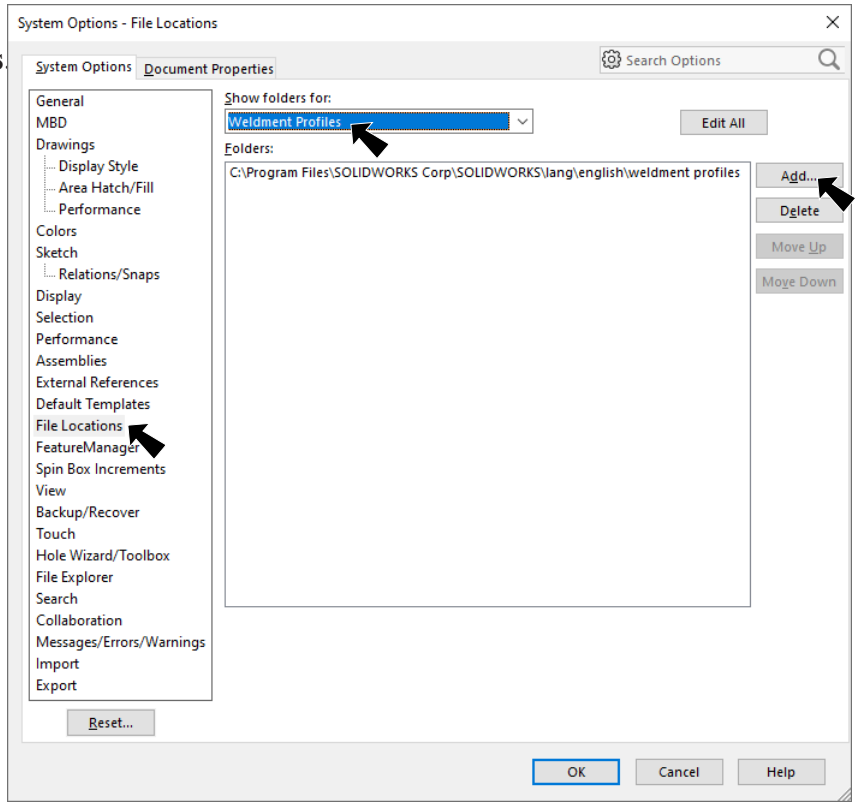


Fig. 16

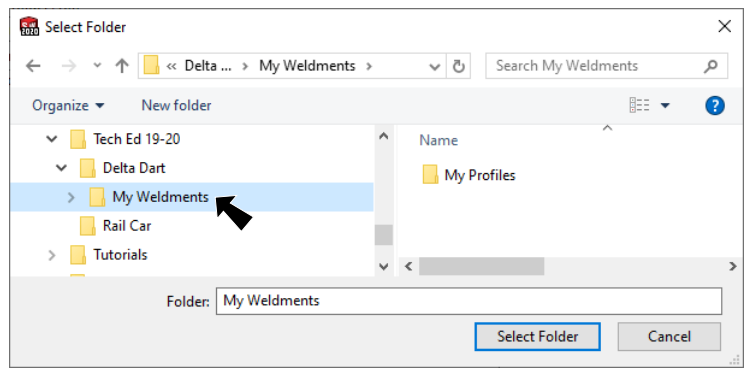


Fig. 17

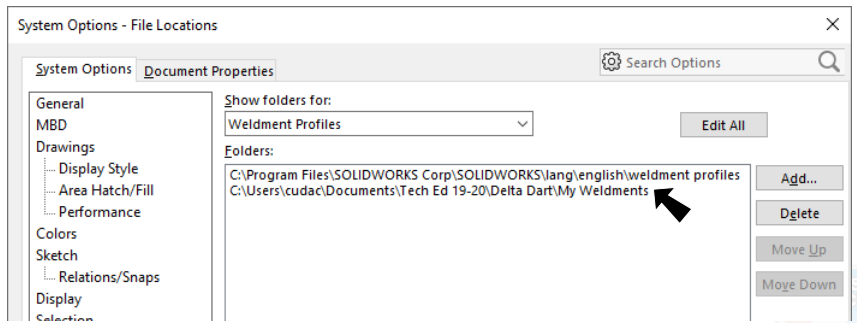


Fig. 18



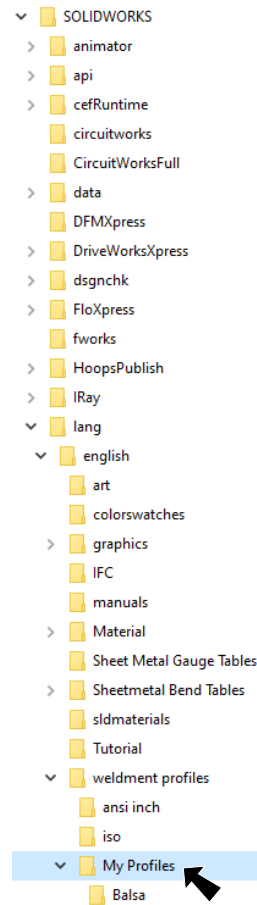
Step 4. If you **have** permissions/rights to your SOLIDWORKS install folder continue here:

Note the path to Weldment Profiles, **Fig. 16**.

C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\lang\english\weldment profiles.

Click Cancel button.

Copy **My Profiles** sub-folder you created earlier into the Weldment Profiles folder in SW install directory, **Fig. 19**.



**Fig. 19**