




## A. Sketch .125 by .125 Rectangle.

- Step 1. Click File Menu > New, click **Part** and OK.
- Step 2. Click **Front Plane**  in the Feature Manager and click **Sketch**  from the Content toolbar, **Fig. 1**.
- Step 3. Click **Rectangle**  (S) on the Sketch toolbar.

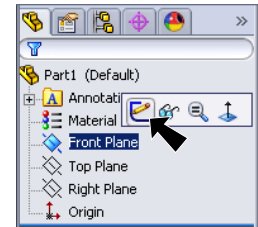



Fig. 1

- Step 4. Draw 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.

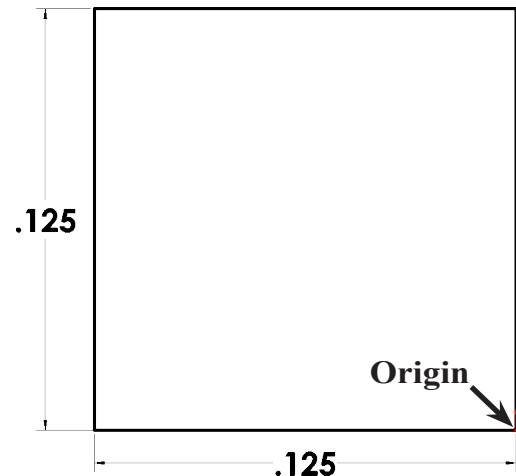




Fig. 2

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

- Step 6. Set dimensions as shown in **Fig. 3**. .125 x .125

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

## B. Points.

- Step 1. Click **Point**  on the Sketch toolbar.
- Step 2. Draw 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.

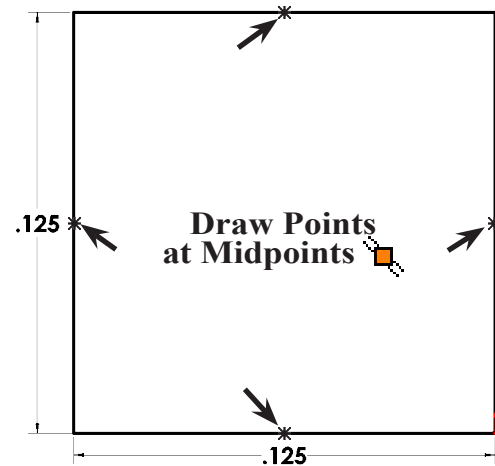



Fig. 3

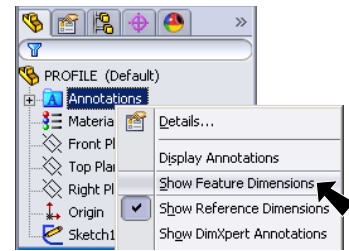
- Step 3. Click **Exit Sketch**  on the Sketch toolbar.

## C. Save as "PROFILE".

- Step 1. Click File Menu > Save As.
- Step 2. Key-in **PROFILE** for the filename and press ENTER.

## D. Properties .125 by 125.

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



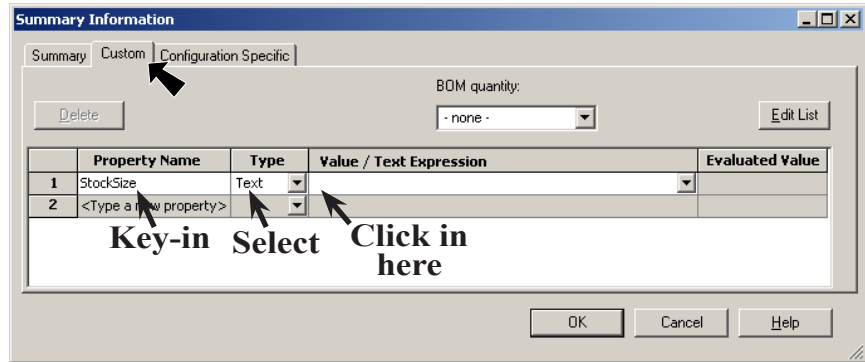
**Fig. 4**

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**

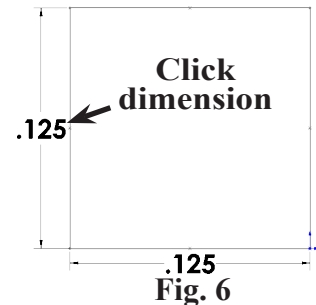
under Value/Text  
Expression  
click in box to  
select the field



**Key-in Select Click in here**

**Fig. 5**

click **.125 height dimension in sketch, Fig. 6**. You might have to move Summary Information dialog out of the way.



**Fig. 6**

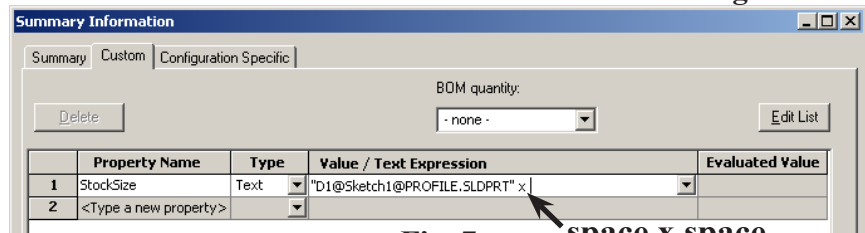
key-in space (spacebar) X space (spacebar), **Fig. 7**

click **.125 width dimension in sketch, Fig. 8**

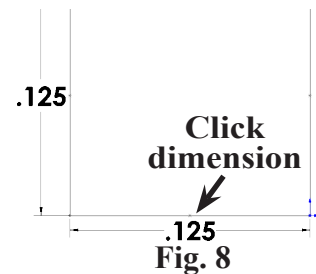
under Property Name (below **StockSize**), **Fig. 9**  
key-in

**Description**  
under Type  
select **Text**

under Value/Text Expression  
key-in **.125in x .125in**



**Fig. 7** space x space



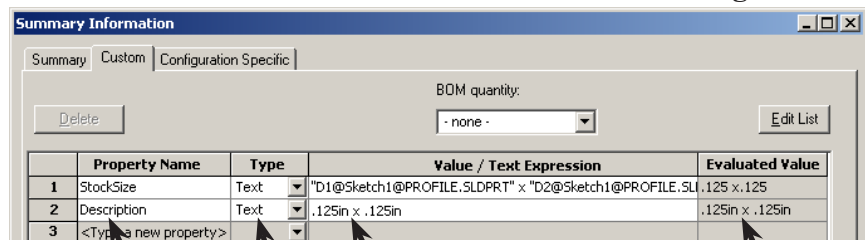
**Fig. 8**

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

**.125 x .125**  
**.125in x .125in**

click OK.

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

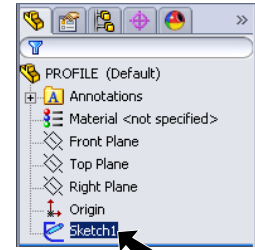


**Key-in Select Key-in**

**Fig. 9**

## E. Save as ".125 X .125" 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.



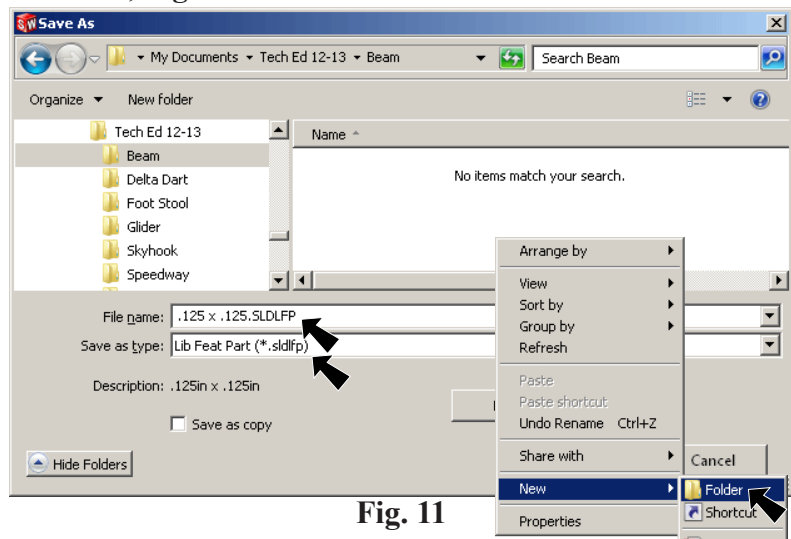
**Fig. 10**

Step 2. Click File Menu > Save As.

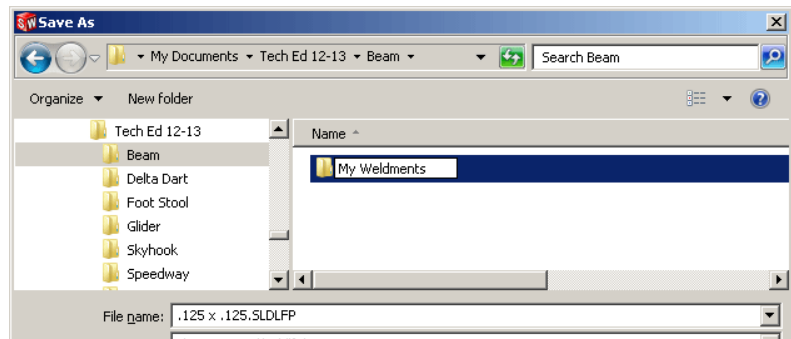
Step 3. In the Save As dialog box:  
key-in **.125 x .125** for file name, **Fig. 11**  
set **Save as Type** to **Lib Feat Part**

**right** click the Name box and click **New > Folder**

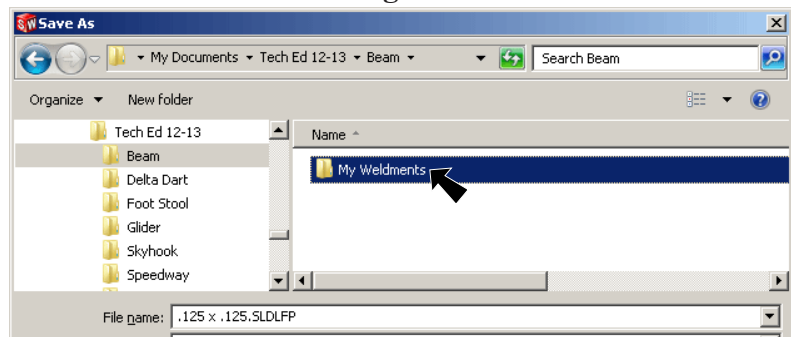
key-in **My Weldments** for folder name, **Fig. 12**  
**double** click **My Weldments** folder, **Fig. 13**



**Fig. 11**



**Fig. 12**

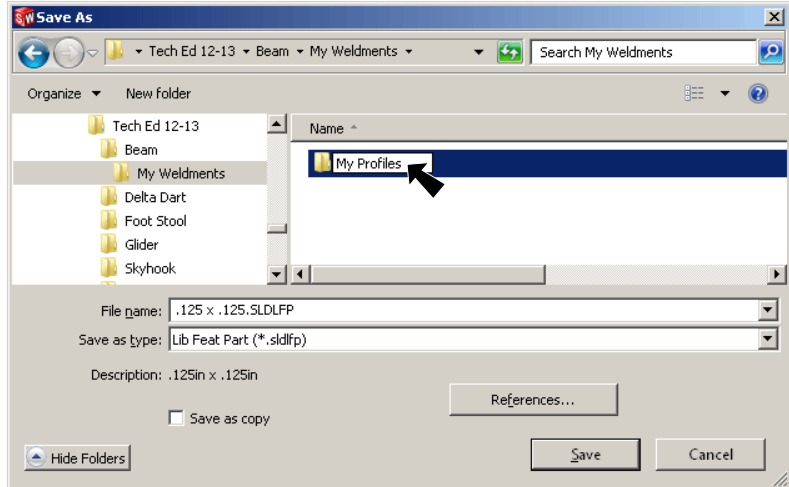


**Fig. 13**

Step 4. **Right click** the Name box and click New > Folder

key-in **My Profiles** for folder name, **Fig. 14**

**double click My Profiles folder**

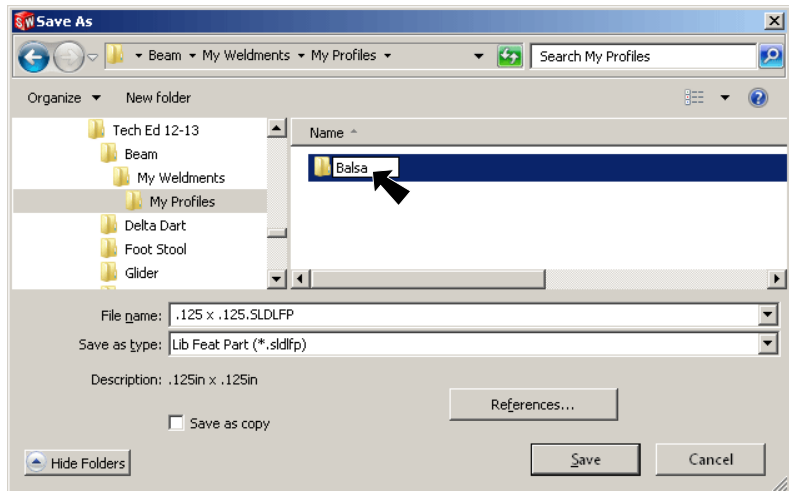


**Fig. 14**

Step 5. **Right click** the Name box and click New > Folder

key-in **Balsa** for folder name, **Fig. 15**

**double click Balsa folder**

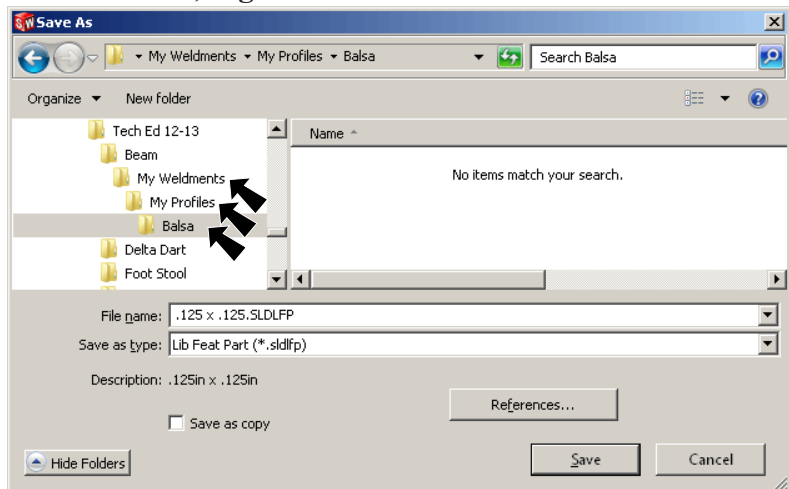


**Fig. 15**

Step 6. Confirm folders: We just created 3 subfolders, **Fig. 16**

My Weldments  
My Profiles  
Balsa

**click Save, Fig. 16.**



**Fig. 16**

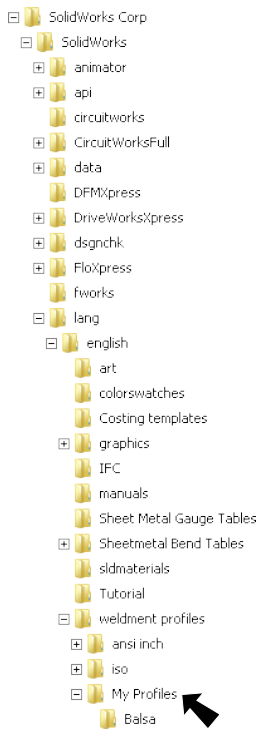
## F. Custom Profile File Location.

Step 1. Click Tools Menu > Options.

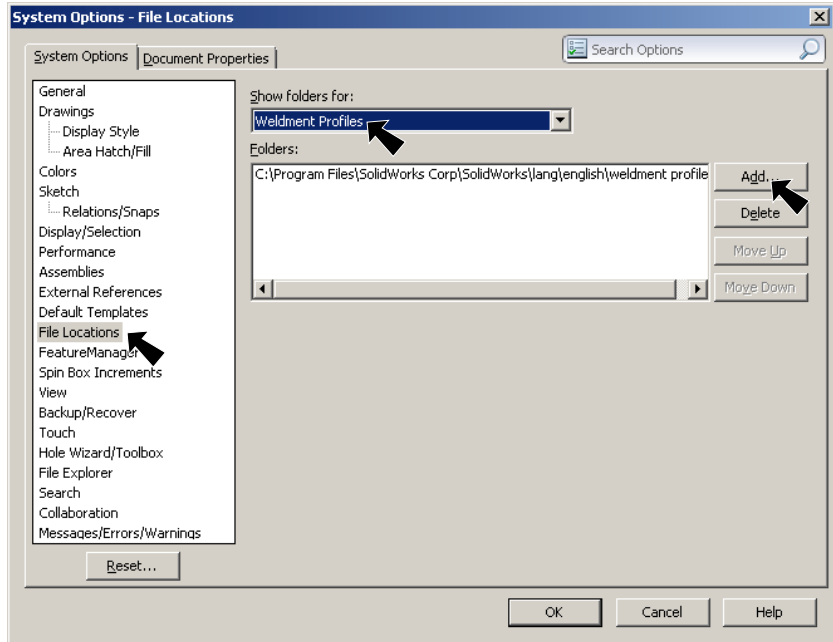
Step 2. In the System Options dialog box, select **File Location** on left, **Fig. 17** under **Show folders for:** select **Weldment Profiles**.

If you **have** permissions/rights to your SolidWorks install folder continue here:  
Note the path to Weldment Profiles.  
Click **Cancel** button.  
Copy **My Profiles** subfolder you created earlier into the Weldment Profiles folder in SW install directory, **Fig. 18**.  
C:\Program Files\SolidWorks Corp\SolidWorks\lang\english\weldment profiles

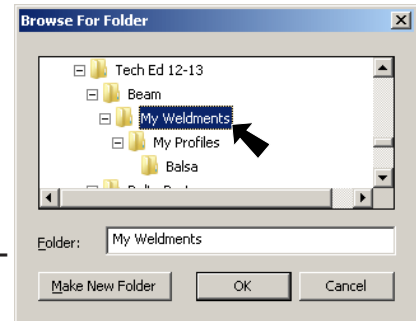
If you **DO NOT** have permissions/rights to your SolidWorks install folder continue here:  
Click **Add** button, **Fig. 17**. **Find and select your My Weldments** folder and click **OK**, **Fig. 19**. Click **OK**, **Fig. 20**.



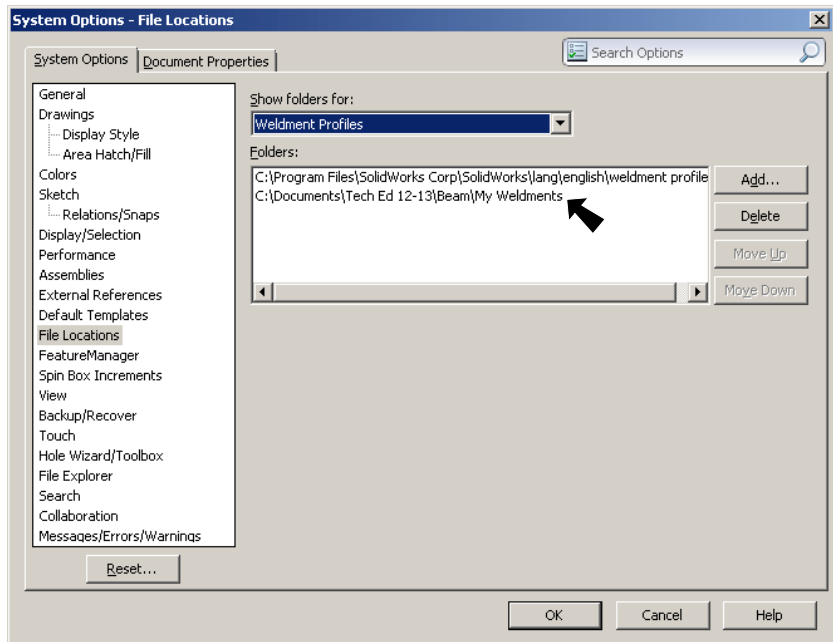
**Fig. 18**



**Fig. 17**



**Fig. 19**



**Fig. 20**