



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**  on the content toolbar, **Fig. 1**.

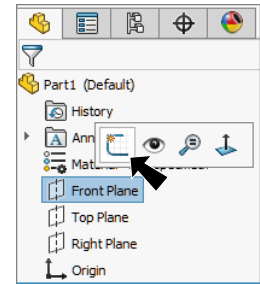


Fig. 1

Step 3. Click **Corner Rectangle**  in the **Rectangle flyout**  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. 2**. .125 x .125

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  and you click.

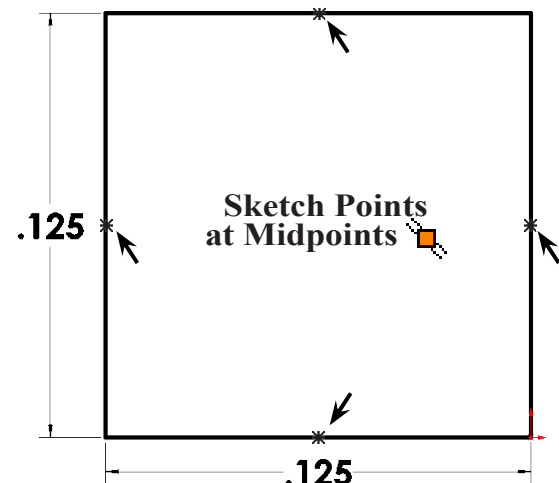


Fig. 3

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**.

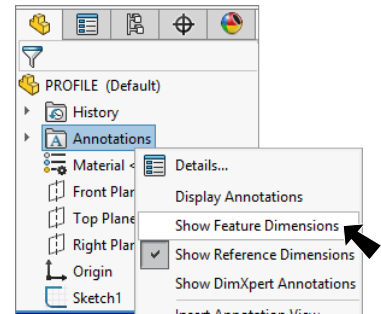


Fig. 4

Step 2. Click File Menu > Properties.

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

under Property Name, **Fig. 5**

select **StockSize**
under Type
select **Text**

under Value/Text
Expression

click in box to
select the field

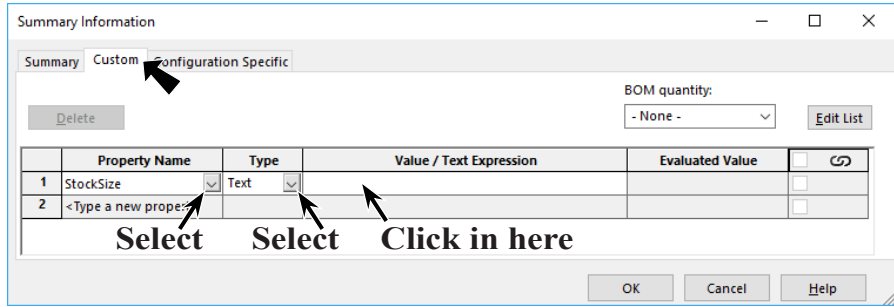


Fig. 5

click **.125 height**

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

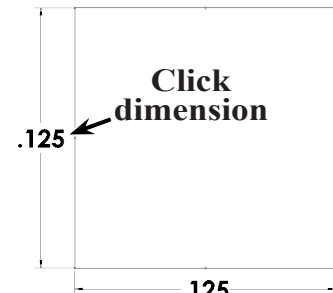


Fig. 6

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

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

under Property
Name (below
StockSize),
Fig. 9

select
Description

under Type
select **Text**

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

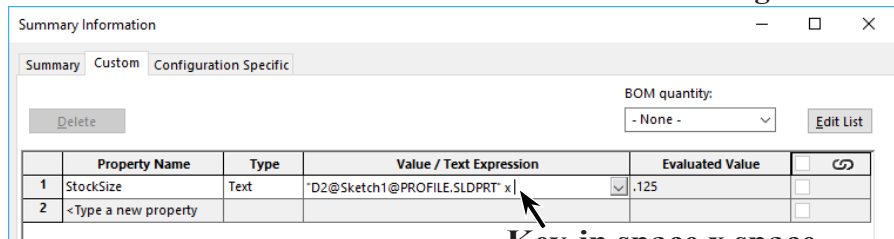


Fig. 7 Key-in space x space

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

.125 x .125

.125in x .125in

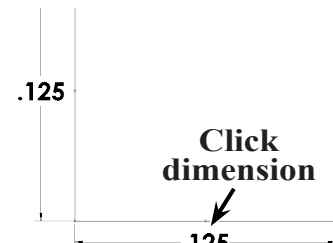


Fig. 8

click **OK**.

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

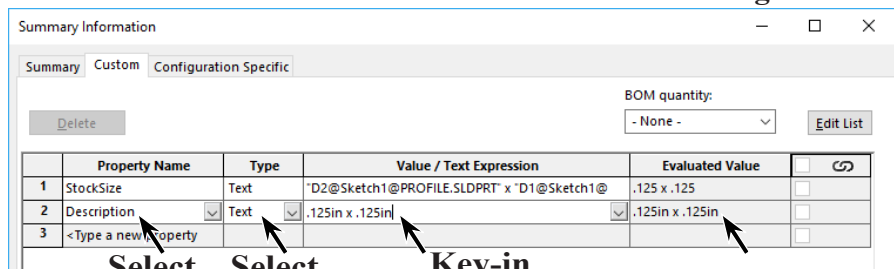


Fig. 9

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.

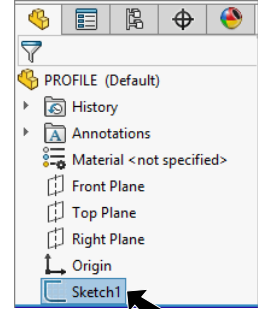


Fig. 10

Step 2. Click File Menu > Save As.

Step 3. In the Save As dialog box, **Fig. 11**
key-in **.125 x .125** for file name
set **Save as Type** to **Lib Feat Part**
navigate to **My Documents/Tech Ed 17-18**

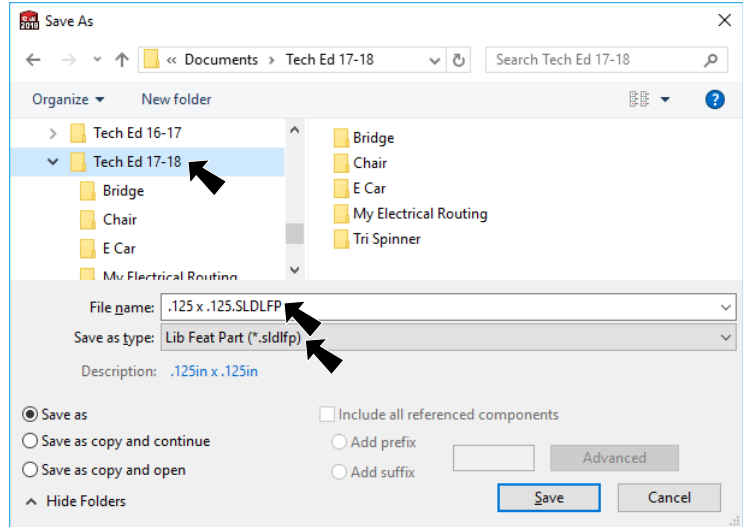


Fig. 11

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

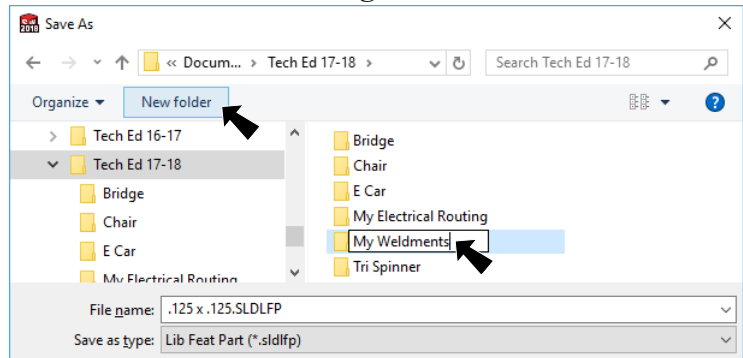


Fig. 12

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

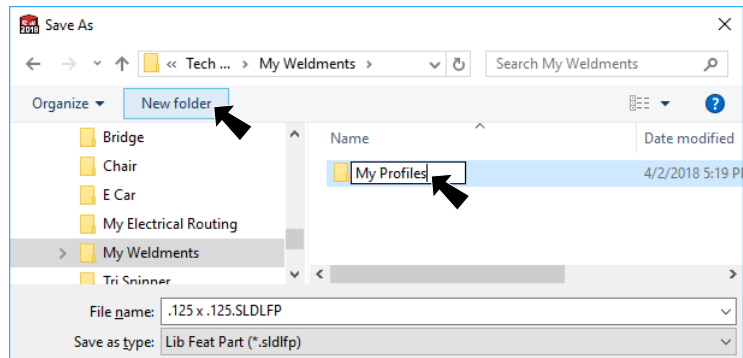


Fig. 13

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

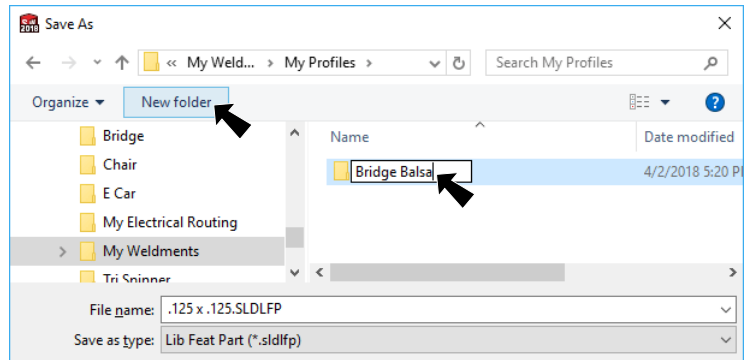


Fig. 14

Step 7. Confirm folders. We just created
3 subfolders in the
Tech Ed 17-18 folder, **Fig. 15**:
My Weldments
My Profiles
Bridge Balsa
and then save .125 x .125.
SLDFP into Bridge Balsa folder.

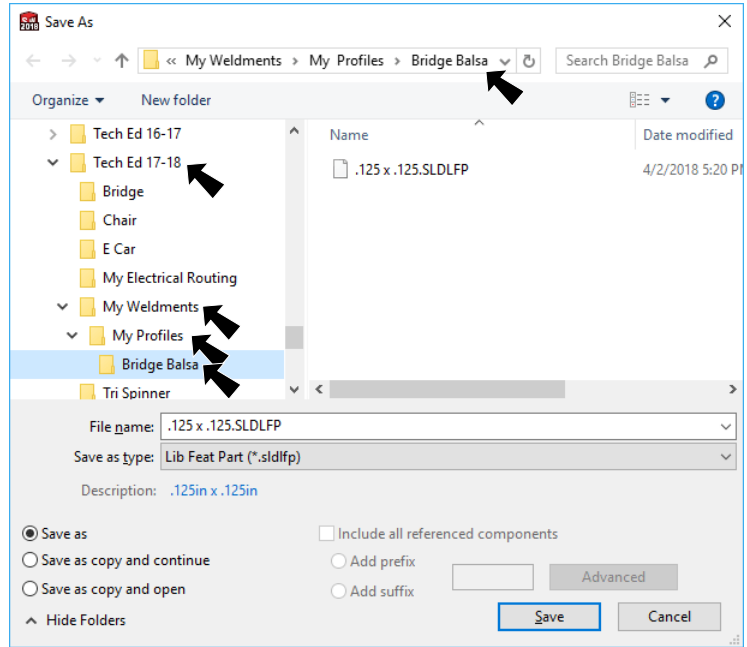



Fig. 15

F. .125 by .25.

Step 1. Change **width dimension to .25**,
Fig. 16. .125 x .25

Step 2. Click **Rebuild**  in the Standard toolbar. (Ctrl-B)

Step 3. Click File Menu > Properties.

Step 4. In the Summary Information dialog box on the Custom tab set:

change **Description Value/Text Expression width** from .125in to **.25in**, Fig. 17

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

.125 x .25

.125in x .25in

click OK.

Step 5. Click **Sketch1** in the Feature Manager to select the sketch,
Fig. 18. The Sketch must be selected when you save as Lib Feat.

Step 6. Click File Menu > Save As.

Step 7. In the Save As dialog box:
 key-in **.125 x .25** for file name
 set **Save as Type** to **Lib Feat Part**, Fig. 19
 click Save.

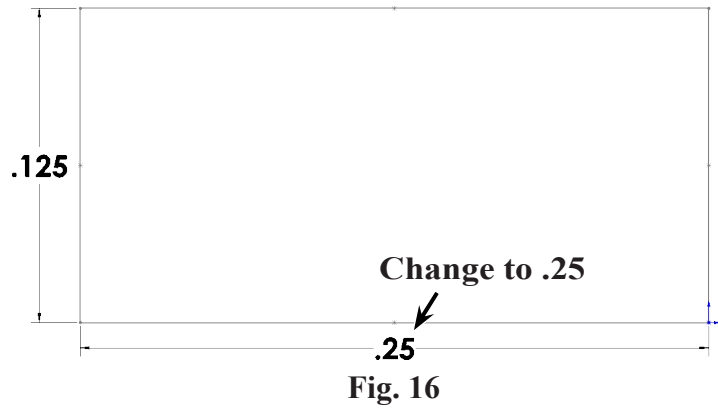
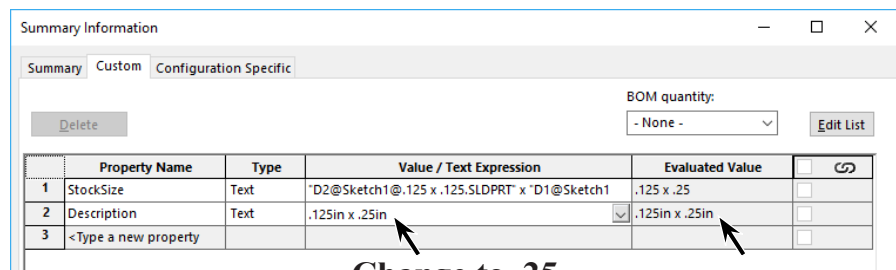


Fig. 16



Change to .25
 Fig. 17

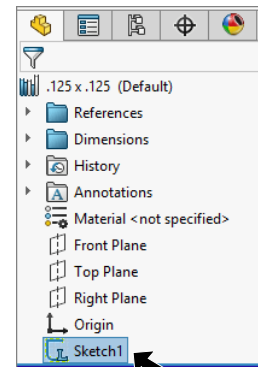


Fig. 18

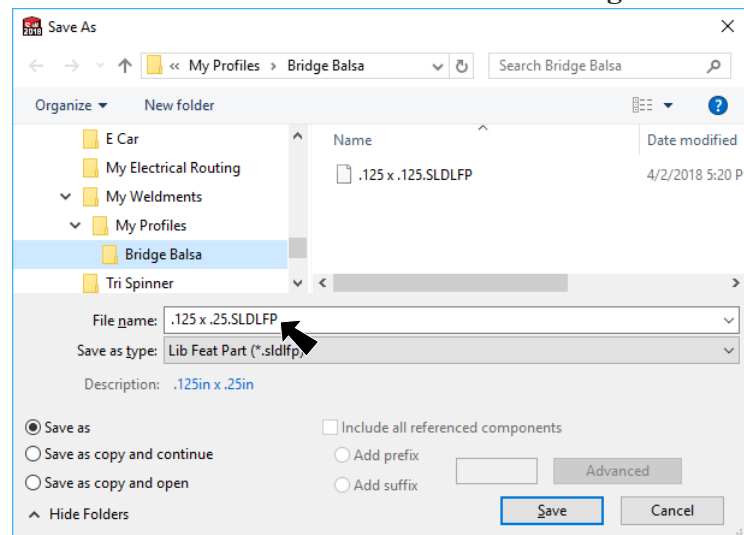


Fig. 19

G. Custom Profile File Location.

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

Step 2. In the System Options dialog box, **Fig. 20** select **File Location** on left 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. 20**.
Navigate to your **My Weldments** folder, **Fig. 21**.
Click **Select Folder**.
Click **OK**.

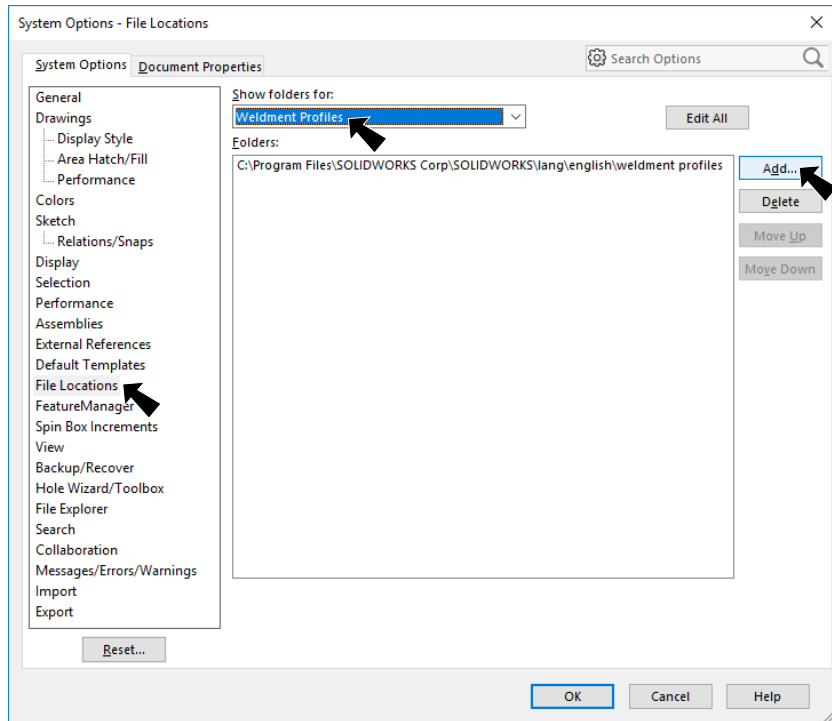


Fig. 20

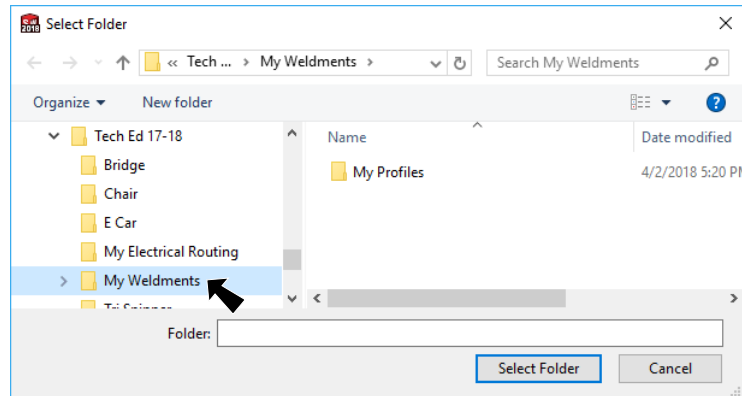


Fig. 21

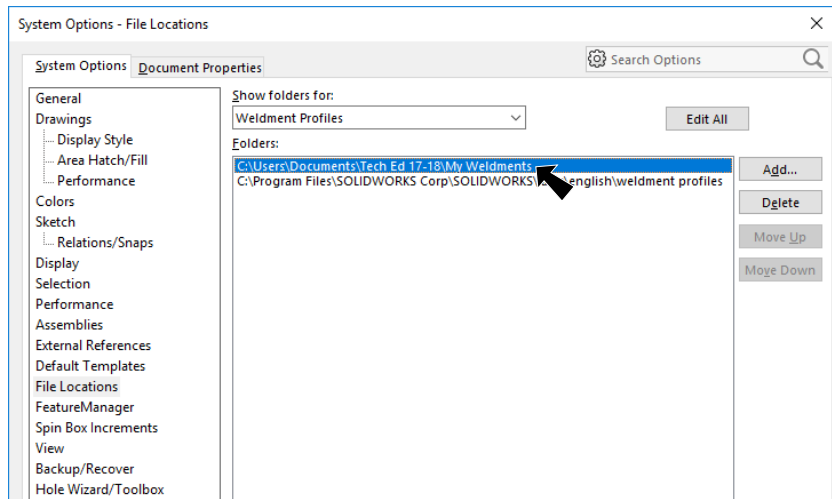


Fig. 22

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

Note the path to Weldment Profiles, **Fig. 20**.

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. 23**.

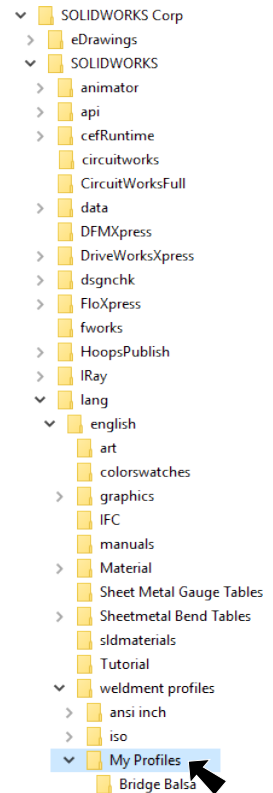


Fig. 23