# LIBRARY FEATURES IN SOLIDWORKS

A library feature is a command which allows you to reuse geometry in a variety of different part files. The feature may be a complex sketch that is repeatedly used, or a collection of features (cuts, bosses etc.) that create standard fittings in your models. The ability to use a library feature can dramatically speed up modelling time in SolidWorks.

**Considerations**

**1- What is the library feature going to be?**

**a. Part?**

**b. Sketch?**

**c. Mixture**

**2- What references shall be used to define the feature?**

**a. Dimensions?**

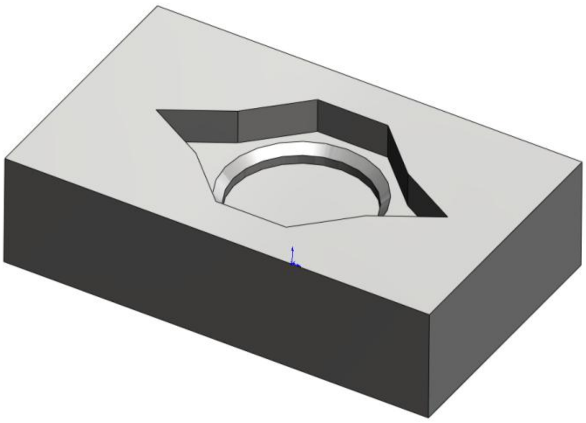
**b. Sketch constraints?**

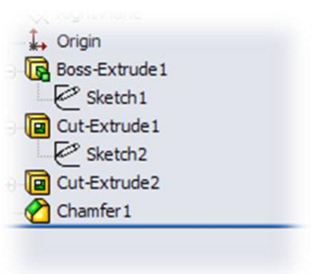
**c. Faces/ Edges/ Vertices?**

## CREATING THE FEATURE

**STEP 1- Create the feature manually in the defining part**

**Design the feature as you would if you were creating it in real life. The constraints and dimensions you place will govern how you locate the feature in future models. For example if you add a dimension to an external edge, you will have to pick a corresponding edge in the future destination model.**

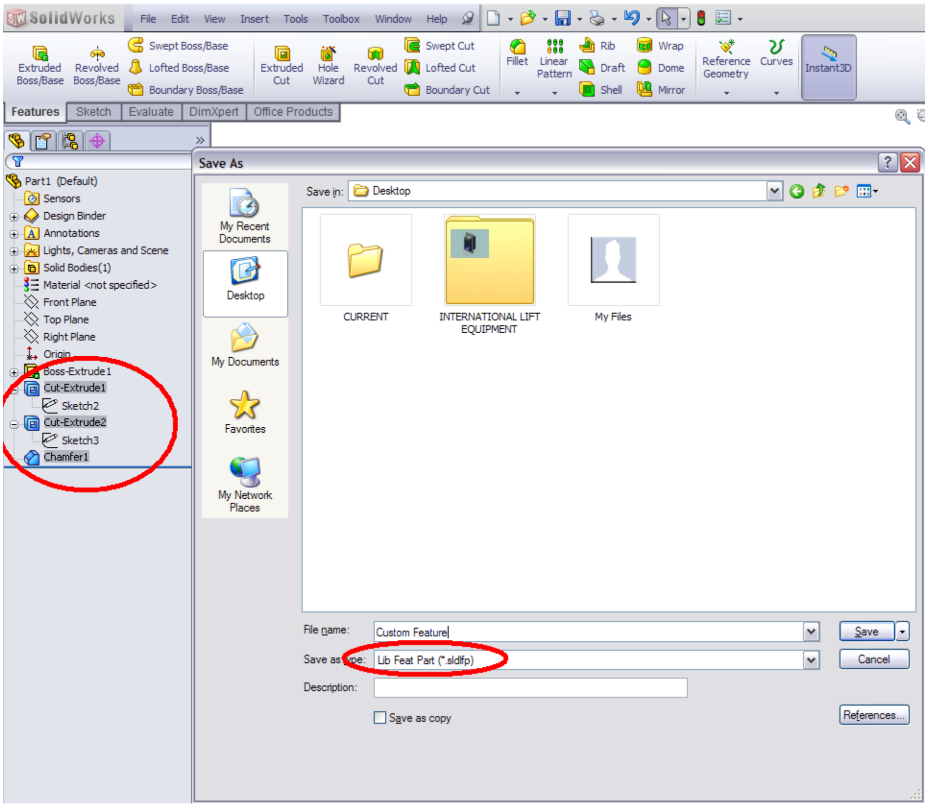





*TIP- if you may ever have to realign the feature i.e. by 90 degrees, try and avoid using constraints such as **Vertical** and **Horizontal** in the defining sketch. These will conform to the global coordinate system of a model and there won't allow a 'vertical' line to be aligned horizontal. Try and use Parallel, Perpendicular and Collinear and these will adjust according to the linked geometry.

**STEP 2- Save the Features**

You must select all feature you wish to reuse before saving. Use the CTRL key and select the features from the tree. Then use **File > Save As** and change the **“Save As Type”** to **Lib Feat Part (*.sldlfp)**

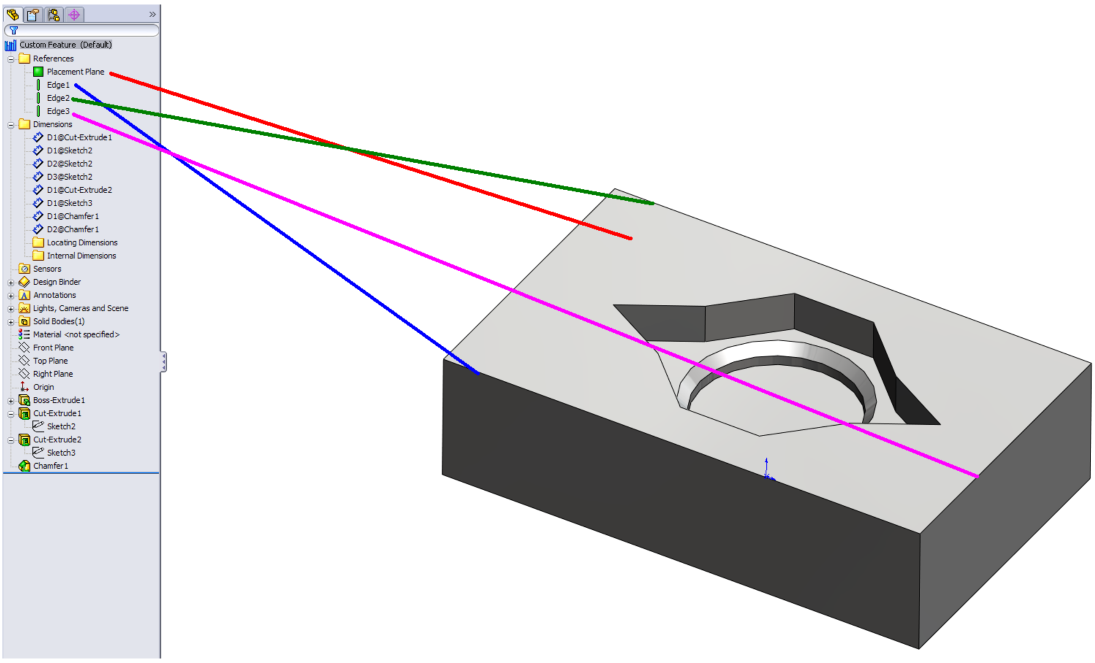



When saved you will notice changes in the Feature Tree. Notably two folders are created called **References**, and **Dimensions**.

**References-** hold the items that will need to be selected in a future model where the library feature is inserted. These are generated because the defining library feature somehow was related to them (sketch on face, dimension to edge etc)

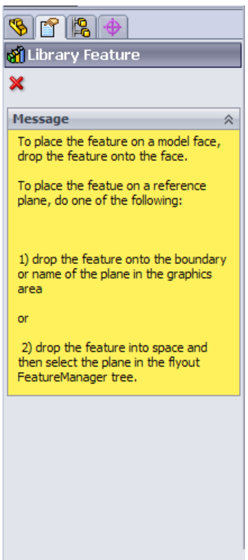
**Dimensions-** these contain all dimensions in the library feature and can be group into **Internal dimensions** (how big the feature is) and **Locating Dimensions** (where the feature is eventually placed. Drag the dimensions into their respective folder and then save the library feature again.

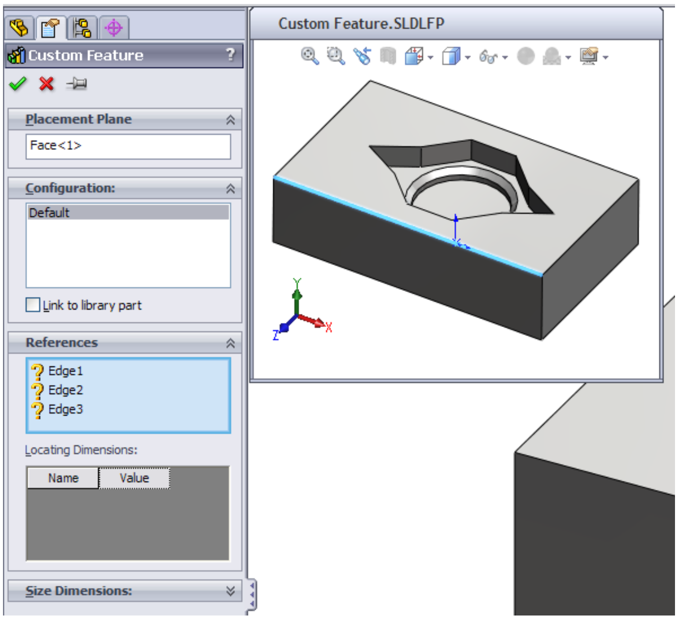
The tree also places a green letter L over any feature that is saved as a library feature



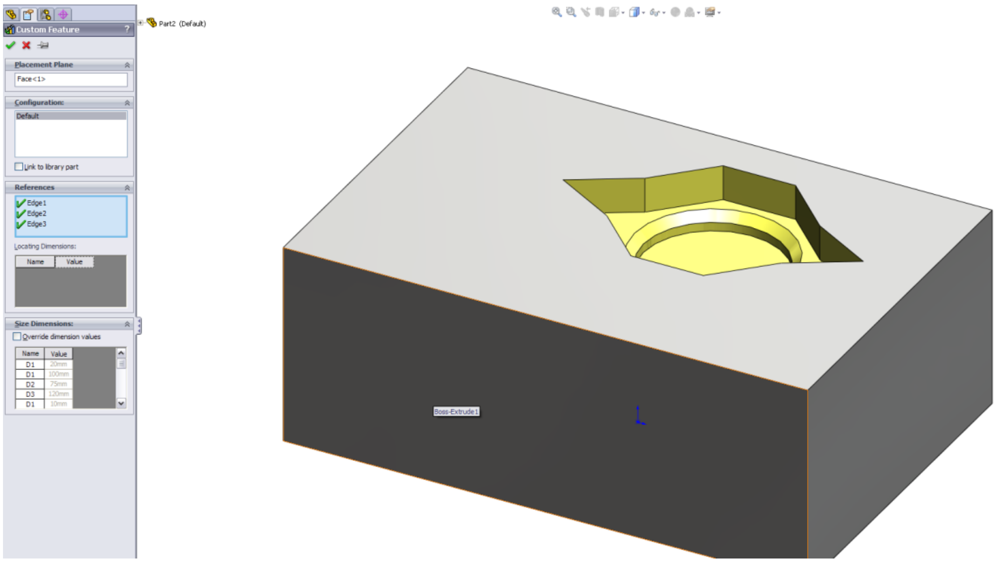

**STEP 3- Reusing the Feature**

To reuse you can drag and drop the feature from Windows Explorer or from the Design Library on the right of the screen. When dragged in you will see a yellow preview outline and the following window on the left of the screen: When placed in position a new window appears:



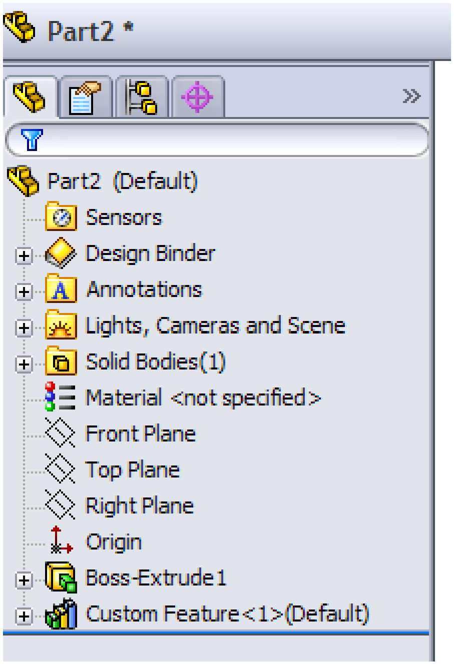



This gives you a preview of the defining part and highlights the references that must be selected on the new part to position things correctly. References marked with a ? still need linking, once linked it turns to a green tick.



When successfully placed the feature(s) turn yellow, and if you specified any locating dimensions you can alter these to position correctly. You can also expand the "**Size Dimensions**" option and override those values to resize the feature.

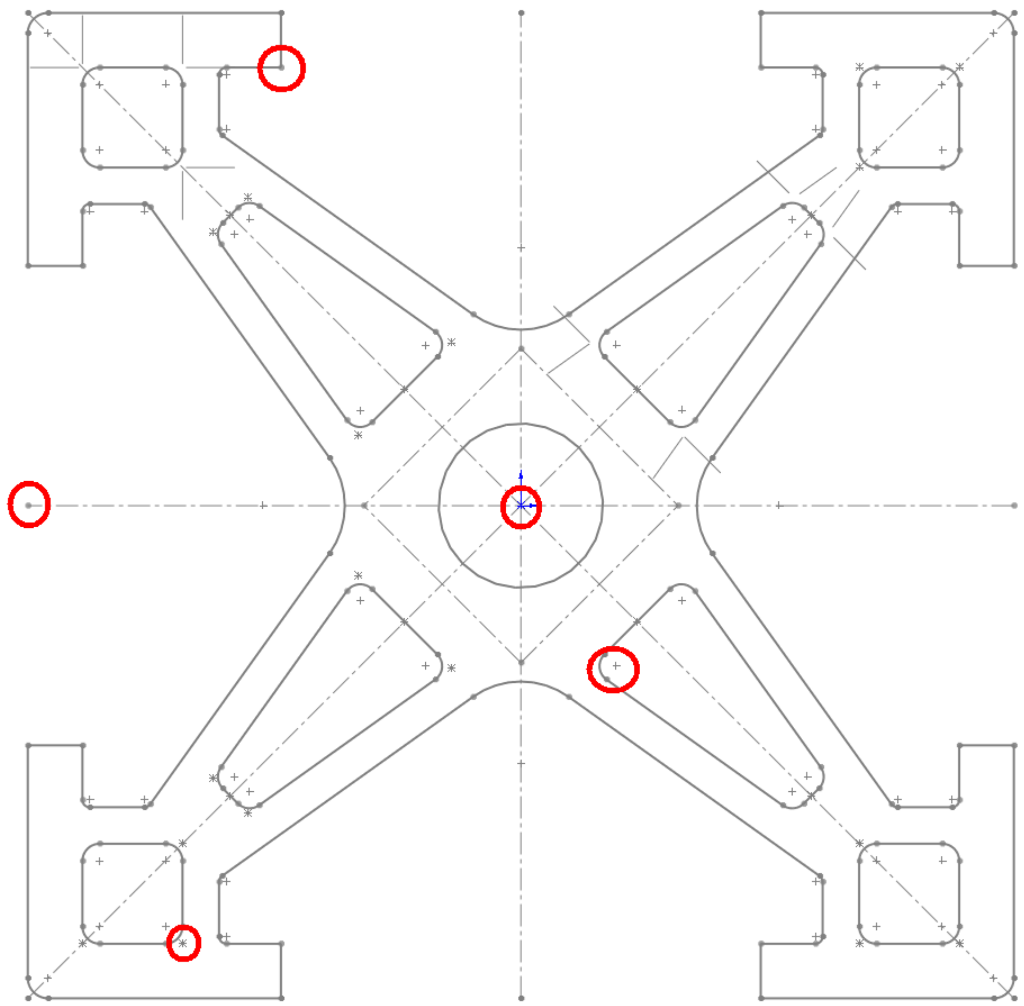
When you confirm the feature they lose their yellow colouring and the tree displays a library icon that groups the features together. You can right mouse click on this and choose to **Dissolve** the feature back to its constituent elements



# Other Library Features

Weldment profiles are a classic example of a library sketch. They are saved in the same way by firstly selecting the sketch and then choosing File > Save As > lib feat part.

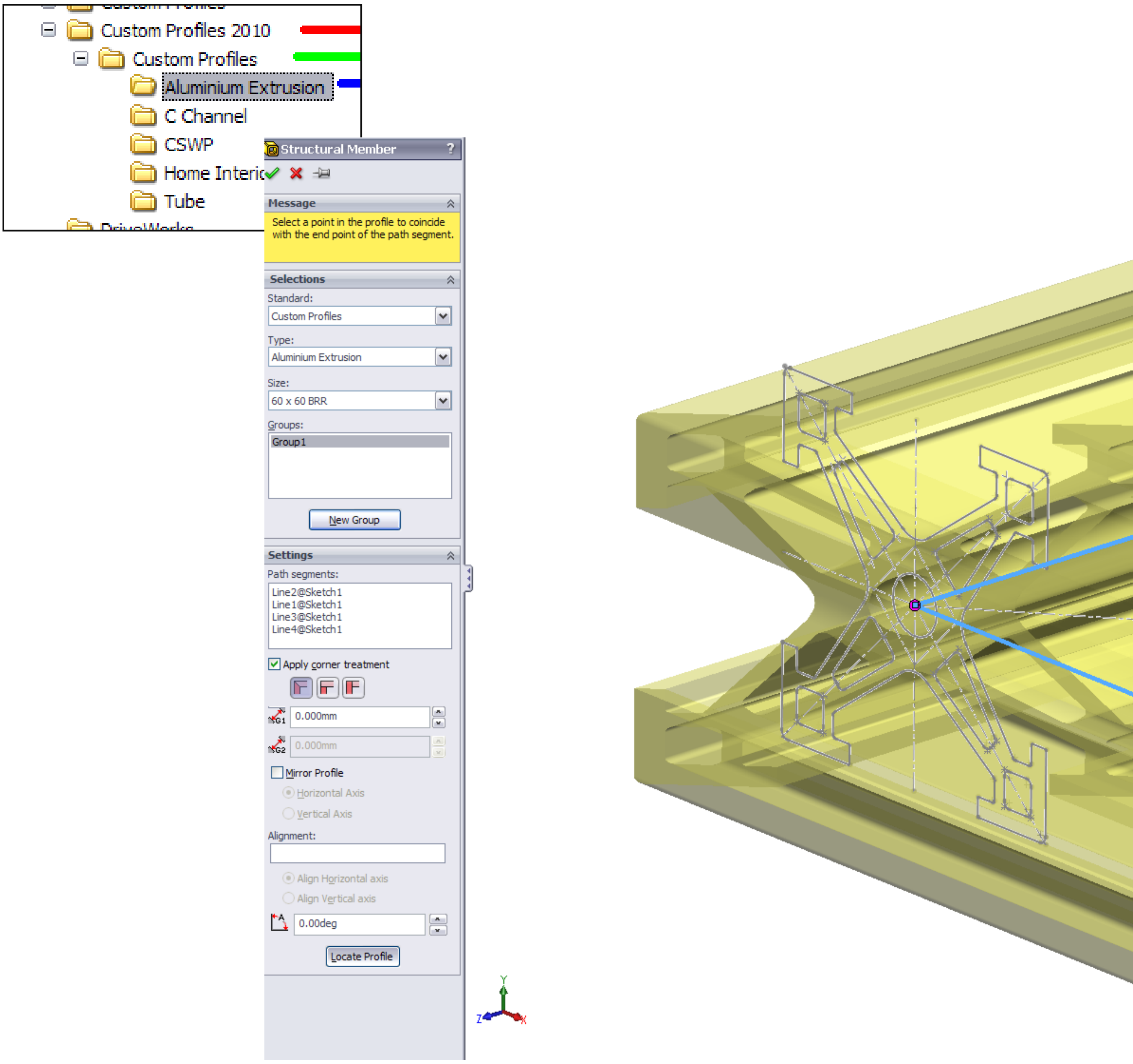
Weldment sketches need to be saved into a specific folder hierarchy as described below and should also contain a number of sketch points or vertices again described below:



The above example is a 2D sketch will can be used to create an aluminium extrusion. The circled entities are either-end points of lines, centres of arcs, virtual sharps of fillets and also manually created sketch points. Why so many? Well the more you have, the more ways you can locate this profile onto its desired path.

Saving the profile as outlined above needs you to firstly pre-select the sketch and then save as a library feature part. You also need to save a weldment profile into a specific location made up of a special folder structure. The image over the page explains why:

the folder structure must follow three levels: Level 1- Red is your Profile Standard. Level 2- Green is the Type and Level 3- Blue is the profile size.




Therefore when you use the weldment feature **Structural Member** to sweep the profile along a path you select from these three folders using corresponding pull down menus. All of the sketch points previously mentioned allow you to located the profile on any of these points as the one that pierces the path. IN the image above right this is the centre point but can be changed with the button “**Locate Profile**”