# Solidworks department handbook

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This handbook should help all designers in the R&D department to work in a common way. If everybody follows the rules it makes it much easier to maintain our specifications.

## 1 Parts

### 1.1 How to start a part – first sketch

You should think about the best plane to create the first sketch. Consider the orientation the part should have in assemblies.

Very often you start with a rectangle or another symmetric sketch. Think about the best position to the origin. If the part is symmetric (like most of the chain links) consider making half of the part and mirror it with a main plane as late as possible.

Use constrains and construction lines to place the sketch on the origin. Don’t use dimensions for this!

Center rectangle on origin:

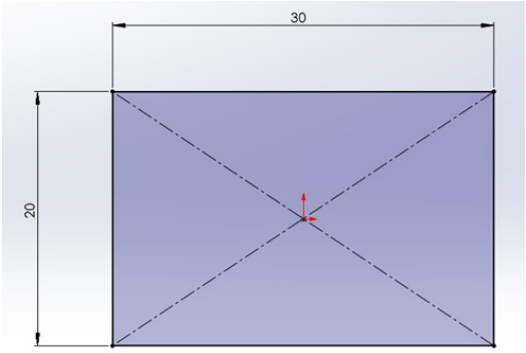



Figure 1 A fully defined sketch.

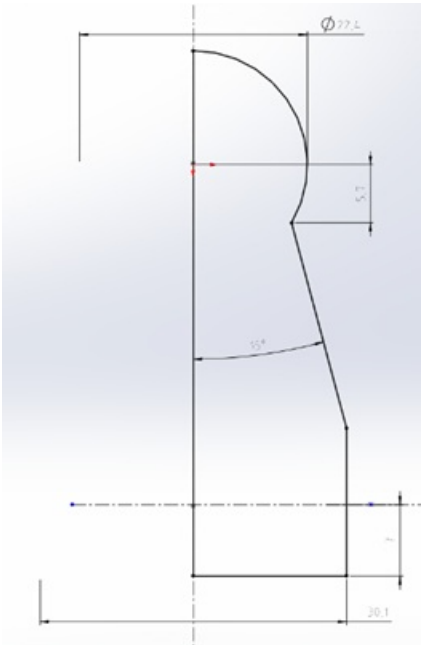


Figure 1 B half base part of link

All sketches should be fully defined!

### 1.2 Working in general

Don’t make too complex sketches. It can be hard to change them later.

Use extra planes and axes if needed. Name them in a logical manner – this will help when making changes later.

Do not use spline when creating aluminum extrusions – most suppliers can not handle them and will replace them with a series of radii.

Try to avoid using spline when making other models also – there is a high risk of trouble during later revisions.

Make radii and chamfers as 3D operations, avoid having them in the sketch if not needed.

When making models for casting, injection molding and extrusions – remember that the tool maker probably will go for symmetrical tolerances. They might change the 3D-model or ask you to do so. Try to have a model that can cope with changes of critical nominal dimensions.

### 1.3 Comments and information

Use comments if needed; to explain the intent of the part, critical features, to explain changes you have made to an older design etc. Bear in mind that someone else will later take over and continue where you left off.

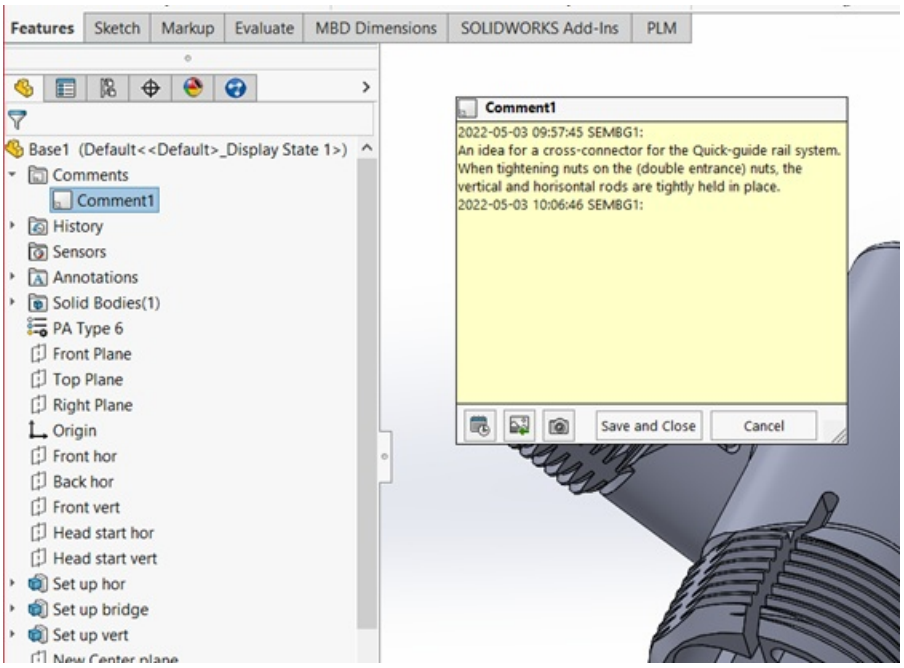



Figure 2 Using Comments can be helpful.

Markup is a graphic alternative or a complement to Comments

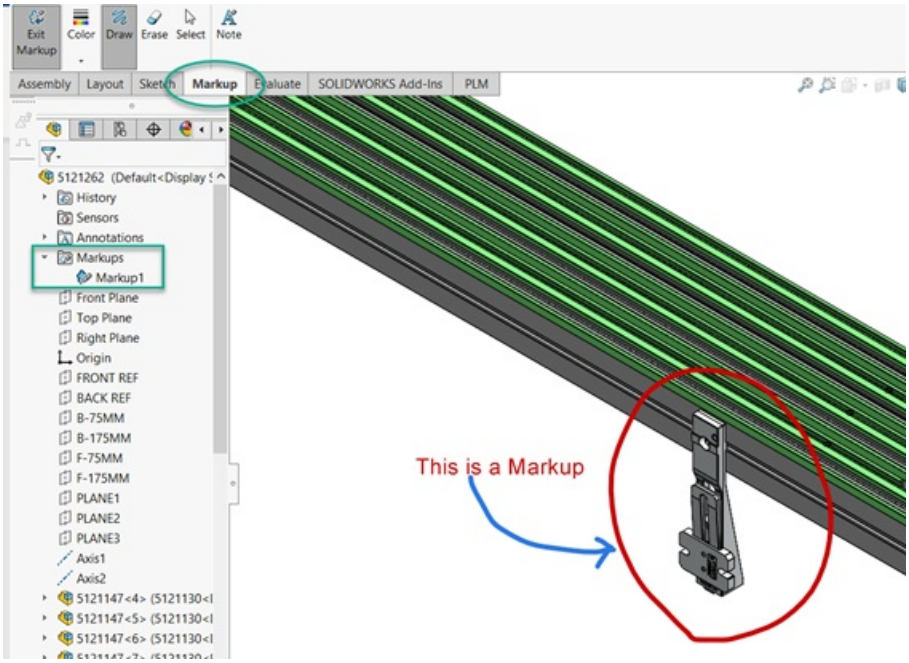


Figure 3 Using Markups also can be helpful.

### 1.3 Set material and appearance

It’s mandatory to set the material of a new part to easily get the weight of the part. It’s needed for the net weight in Coral.

The appearance should also be set to prepare the part for render images. Preferably use the predefined FlexLink appearances, see paragraph 5 Rendering. Avoid setting strange colors if not needed.

If you revise a part, please set both if missing.

### 1.4 Name feature tree items

At least important features in the tree should have names to easily find them. E.g. „Top plate“.

### 1.5 Combine radii and chamfers

It’s recommended to combine as many edges as possible to create radii or chamfers to keep the feature tree short. Create them as late as possible to avoid bad parent-child relations (see 1.6).

Name them e.g. “Radius R1”.

It is not recommended to combine different radii in one command (multi radius fillet). It would make it difficult for other colleagues to change it.

### 1.6 Parents and children

Using parent and child relations between features can cause problems when later making changes to an early sketch. Using reference planes can be an alternative, if done with care. Some features will always be dependent on each other, while other relations may just cause problems later on.

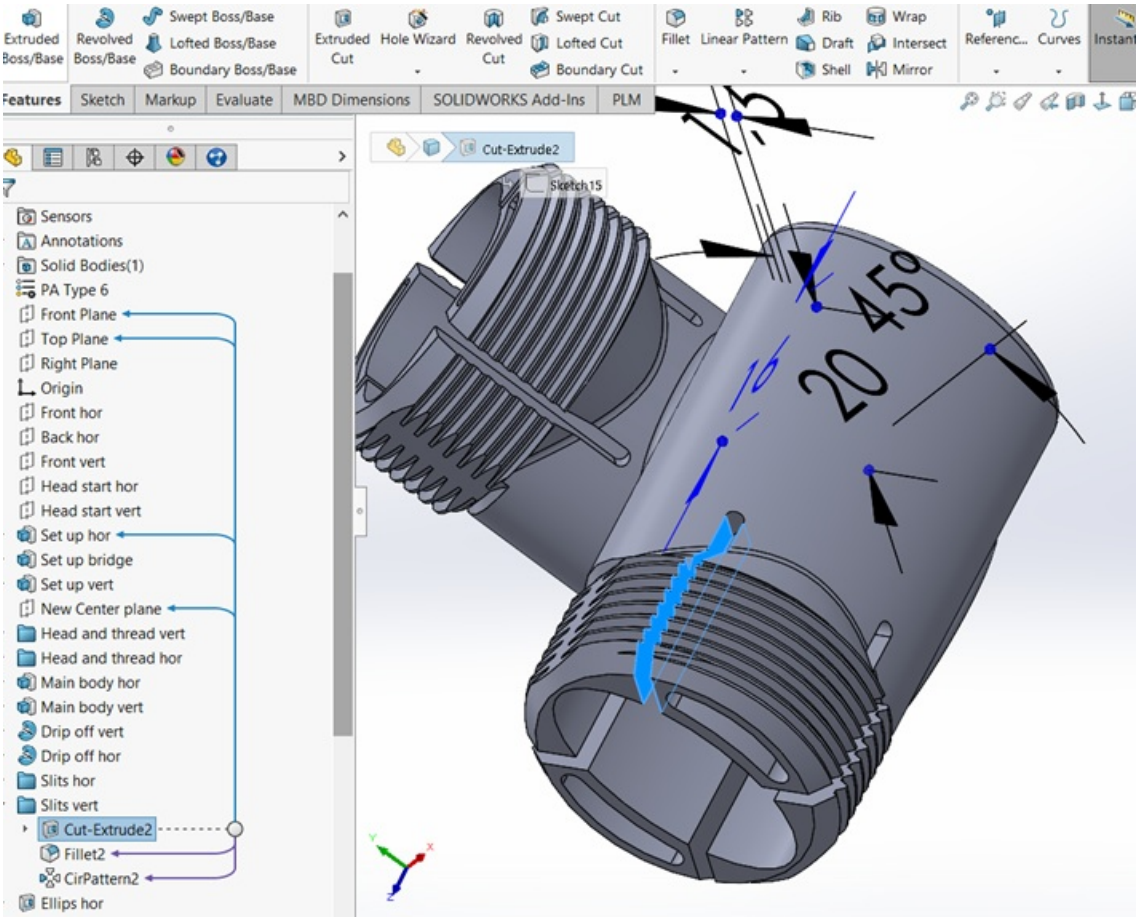



Figure 3 A feature with few dependencies, all features have names to help understanding/finding them in the tree.

### 1.7 Special parts

#### 1.7.1 Chain links

Make one half only and mirror it as late as possible.

Don’t forget to set all drafts.

Check if the chain behaves as intended in all units (e.g.drive units with slag)

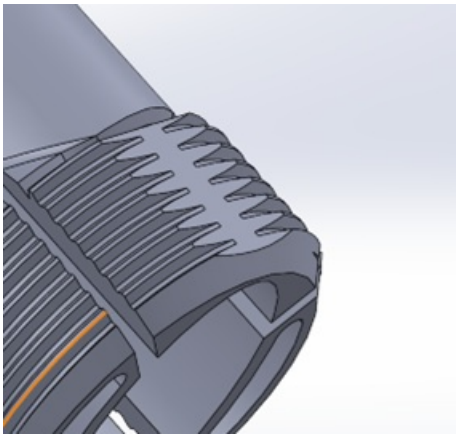
#### 1.7.2 Profiles/beams

Make one half or one quarter only and mirror as late as possible. All features related to the section should be present before mirroring – except small corner radii or fillets.

#### 1.7.3 Other molded or casted parts

Don’t forget to set all drafts. When supplier / toolmaker has been selected, confer with them about tool split-lines, injection gates, moving cores, lifters, ejection pins etc. to get drafts correct. “Rough” surfaces and some materials need added draft.

Consider the split-line of the tool; the risk of flash (excess material) or damages in the tool.



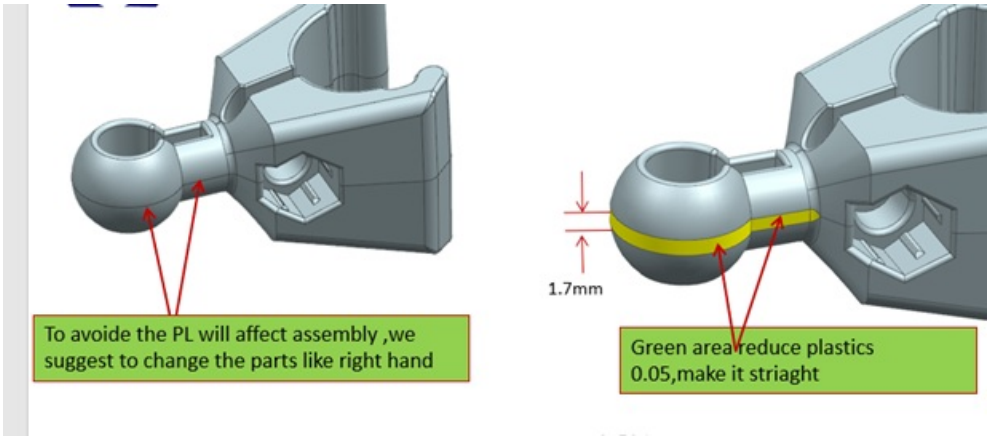



Figure 4 Partially removed material at split-line, to ensure function even if flash occurs.

### 1.7.4 New standard parts

If you need to create a new standard part, do it in a good way. E.g., screws should have a cosmetic thread with the proper length, material and appearance. Do not make a thread with a 3D operation if not needed. The part could slow down big assemblies.

If you revise old standard parts update at least material and appearance.

### 1.7.5 Re-use of a part

When creating a second generation, or a variant, of an already existing component/part it is important to retain all vital information. Keep the original planes, remove any references to other parts, do not import bodies – keep it simple and easy to modify.

Also, when doing such a work, you will probably encounter problems understanding original intent, build-up sequence etc. – especially if you didn´t create the original version. Take ½ extra hour to “clean up” and clarify the feature tree, naming features etc. if you can find the time, saving the time you have spent for the next guy/gal.

## 2 Assemblies

### 2.1 Insert first part

Don’t simply drop the first part into a new, empty assembly. If feasible, the origin of the part

should be mated to the origin of the assembly. Depending on the geometry of the part the main planes are in the middle then.

### 2.2 Insert next parts

If possible, the next parts should be mated to the origin or the main planes of the assembly.

### 2.3 Feature tree and BOMs

The parts in the feature tree should be arranged in the order they shall appear in the Coral BOM. The recommended order is:

- FlexLink parts (FLX… numbers)
- External parts (270… numbers)
- Standard parts (screws etc.)

If it’s done like this the balloons will match the automatically generated BOM in Coral.

Identical standard part should be grouped together.

If the assembly contains many screw and other standard parts it’s recommended to create a folder called e.g. „Standard parts“, and move it to the end of the feature tree. The parts will appear in the end of the Coral BOM.

If you add or remove parts in the model, it’s recommended to delete the BOM in Coral and store a new one. Be careful if it contains manually added items - this works only if all items including manuals etc. are CAD parts in the assembly

### 2.4 Exploded views

Exploded views are good to show more complex assemblies. Use exploded lines where reasonable.

Don’t forget to collapse the assembly before storing it to Coral to get a good preview.

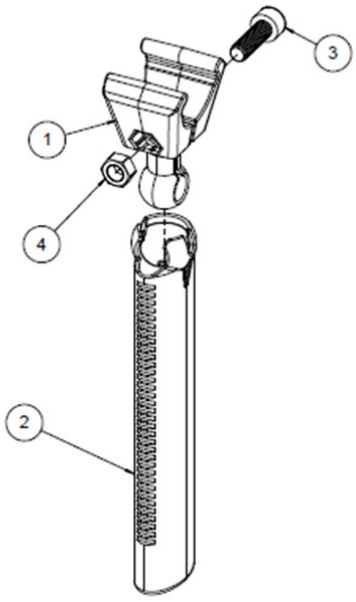



Figure 6 An exploded view, with the balloon numbering in correct order and logical assembly order.

## 3 Drawings

### 3.1 Layout

**LINK**

### 3.2 Geometrical tolerances

**LINK**

### 3.3 Balloons and BOM

Prepare the feature tree as described in 2.3. Insert a BOM table into the drawing to be able to place intelligent balloons connected to the CAD model. After placing all balloons hide the table.

It should be avoided to set the balloons to text and edit them.

### 3.4 Information on the drawing

Remember to specify surface roughness (VDI), and especially important dimensions etc. (Quality measurement).

If the 3D-model is the main source of information, reference to this file should be made from the drawing. General tolerance for dimensions and shape should be stated.

If the material is polymeric it might be needed to state a “setting time”, after which dimensions are valid (normally 24 hours).

For molded parts, maximum allowed flash should be stated.

## 4 Coral and Solidworks

### 4.1 Error messages

**LINK**

## 5 Rendering

### 5.1 FlexLink settings

FlexLink has prepared scenes and material appearances. To be able to use these you first have to include these “libraries” into SolidWorks, via the “file locations” under Options – see the computer settings chapter on how to do this:

**LINK**

Then apply the scenes and appearances using the guide:

**LINK**